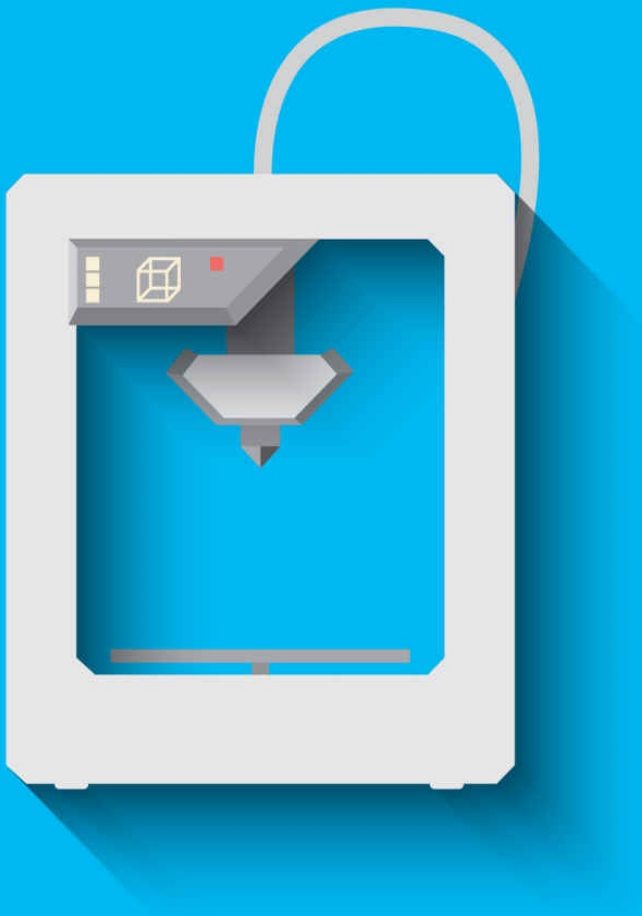


2022 EDITION

3D PRINTING FAILURES

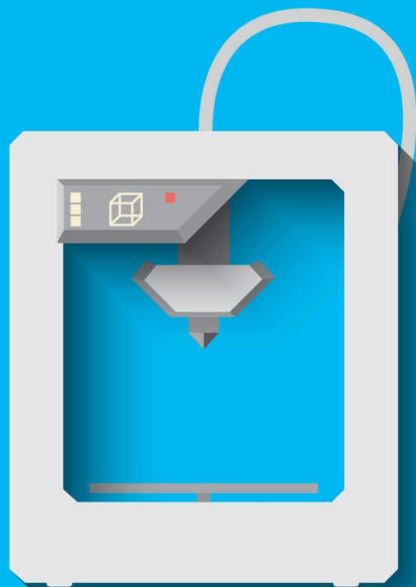


HOW TO DIAGNOSE AND
REPAIR ALL DESKTOP 3D
PRINTING ISSUES

SEAN ARANDA

2022 EDITION

3D PRINTING FAILURES



HOW TO DIAGNOSE AND
REPAIR ALL DESKTOP 3D
PRINTING ISSUES

SEAN ARANDA

Table of Contents

[Introduction](#)

[Good Practices](#)

[Diagram of a 3D Printer](#)

[Electrical Safety](#)

[Mandatory Maintenance](#)

[Material Science](#)

[Materials and their Settings](#)

[Quality Options](#)

[Limitations with 3D Printing](#)

[Speed Limitations](#)

[Cura Tricks](#)

[Important Accessories and Replacements](#)

[New Innovations](#)

[Post-Processing](#)

[Upgrades & Purchasing a New Printer](#)

[Resin Printing](#)

[Diagnosing Failures](#)

[Bed Adhesion](#)

[Build Plate Not Heating](#)

[Build Plate Not Reading Correct Temperature](#)

[Built Up Material on Nozzle](#)

[Curling of Layers and Angles](#)

[Elephant Foot](#)

[Extruder Motor Skipping](#)

[Filament Snaps](#)

[Gaps in Walls](#)

[Gaps on Top Layers](#)

[Ghosting](#)

[Hotend Can't Reach or Maintain a Temperature](#)

[Hotend Not Heating](#)

[Hotend Not Reading Correct Temperature](#)

[Layer Bulges](#)

[Layer Shifts](#)

[Missing Layers and Holes in Prints](#)

[Model Errors](#)

[Not Finding Home and Inverted Prints](#)

[Nozzle Clogs](#)

[Over/Under Extrusion](#)

[Parts Being Knocked Over](#)

[Parts Not Mating Together](#)

[Patterns in Outer Surface](#)

[Poor Layer Adhesion](#)

[Problems with "Power Loss Recovery"](#)

[Running Out of Filament](#)

[Settings Issues](#)

[Squished Layers](#)

[Stepper Motors Overheating or Malfunctioning](#)

[Stringy or Blobby Prints](#)

[Stripped Filament](#)

[Unlevelled Build Plate](#)

[Warping](#)

[Z-Axis Wobble](#)

[Z- Height Calibration](#)

[Tips if Still Not Working](#)

[Resources](#)

[About the Author](#)

Introduction

When I first started working with 3D printers in early 2015, I was overwhelmed by the amount of knowledge required to have consecutive, successful prints. Since I was aware of the acceptable failure rate for most other manufacturing machines, I was blown away by just how inconsistent 3D printing can be without frequent, proper maintenance.

Many desktop 3D printing companies advertise a plug and play machine that can be operated consistently without any engineering background, which is generally not true. 3D printing has been marketed by many as a magical solution to manufacturing and prototyping, but anyone who has used these machines will tell you differently.

It has now been two years since my last edition, and it was time to update all of the content to include new products, features, and failures that I have experienced. Some chapters are very similar with only a few minor tweaks, while others have been completely re-written from scratch. My goal is not only to include new information, but to have this book flow a bit better than previous editions.

This edition will start with a few chapters that can be read in order, flowing more like a standard informational book. Once you hit the “Diagnosing Failures” section, it will read more like a textbook that just requires you to find your specific issue, and go to that particular chapter. The goal of this is to get you familiar with 3D printing and some aspects about it that you might not have thought about. This will then help you to diagnose your failure without ever needing to read the particular chapter on the topic.

These introductory chapters include term explanations, electrical safety, limitations, maintenance, upgrades, innovations, tips, and most importantly – material science. This “Material Science” chapter has valuable knowledge from Nicolas Tokotuu of Polymaker. Nicholas is my go-to for material questions, and he knows his stuff better than anyone I have personally met. This “Material Science” chapter can help you to understand why a problem is happening, which can help immensely with fixing problems before they ever occur. If you were to only read one chapter in this book, that would be it.

If you finish these introductory chapters and come to the “Diagnosing Failures” chapter without having your particular issue resolved, the goal is for you to review the photos and find your issue. Once you find the particular failure you are experiencing, there will be specific page numbers listed below the photo that you will reference to solve your issue.

My goal with this book is to take every failure I have experienced and put it into one resource. This book should be able to help you fix close to 100% of the problems you are going to experience with your 3D printer. If you purchased this book and it does not help you with your specific problem, I offer you to contact me anytime at my YouTube channel “The 3D Print General”, or to email me at Sean@3DPrintGeneral.com. I may take a few business days to respond, but I will do my best to help. I also encourage you to email me with proof of purchase to receive higher-definition colored photos, since the publishing process often reduces the quality.

Warnings for using your 3D printer

Since I personally believe this industry does not do a proper job of warning about the dangers involved with 3D printing when advertising products to average consumers, I feel it necessary to warn you of the real possibility of a fire while operating one of these machines, especially if the machine was not set up properly with “thermal runaway”. This is equipment that draws a lot of power, shakes and moves for hours, and has a lot of wires that can be dislodged or frayed. Many inexpensive manufacturers do not take the proper cautions. You shouldn’t run your printer next to curtains or other flammable things, and you shouldn’t leave your machine printing alone for hours if you are not confident in your build quality. I personally have an AFO Fire Extinguishing Ball mounted above my printers as a preventative measure, and suggest everyone else to do the same.

It is also a good idea to check out the video by Thomas Sanladerer titled “Everything you need to know to make your 3D printer fireproof!” on YouTube. There are some complicated things that he goes over, but it is crucial that you understand this before you purchase a \$200 printer with a heated build plate and leave it unattended. There is a somewhat abridged version of his video as a short chapter in this book titled “Electrical Safety” that you should read over, in which some of the basics are covered.

Another concern is that not much research has been done in relation to the amount of harmful particulates put into the air from melting plastic in this fashion. You can imagine that if you threw some Legos into an oven and inhaled the smoke that was created, you would likely be doing some serious damage to your health. You should make sure that your 3D printer is in a well-ventilated area, and that you do not stand over the hotend while it prints. Some manufacturers have factored this issue into their build design and enclose the machine and ventilate it with filters.

Notes about different types of printers

The vast majority of this book, including all failures in the diagnostic section, are regarding FDM 3D printing. I have added for this 2022 edition a short “Resin Printing” chapter, though I am not an expert in that style of printing, so I am only including my basic knowledge for those brand new to the topic. Please refer to that chapter if you plan on purchasing a resin SLA printer.

What is FDM 3D Printing?

There are quite a few different forms of 3D printing available today, but the most common used in homes around the world is known as Fused Deposition Modeling (FDM). FDM printing works by laying down consecutive layers of material at high temperatures, with each layer given time to cool and bond together before the next layer is deposited.

This can actually be thought of as the inverse of computer numerical cutting (CNC). 3D models are transformed into G-code via a slicing program, which work as instructions for the 3D printer; telling it exactly where to move next and how much volume is required to extrude. This additive process only uses the amount of material required to create the part, versus CNC which is subtractive and requires excess material which it is then cut from. The only exception to this is the support material required for overhangs in FDM 3D printing, acting as a form of scaffolding that is broken off after printing.

When you think about FDM 3D printing, there really are only 3 things that make it unique from other manufacturing processes: the material, the slicing software, and the extrusion process. Motors, boards, G-code movements and everything else are not unique to 3D printing. This is why I consider the “Material Science” chapter so important, along with the “Settings Issues”. It is also why I believe the extruder and hotend can be the two most important upgrades on any 3D printer, more so than anything else.

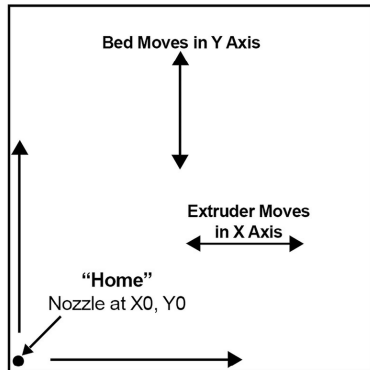
Pros to Using FDM Printing

First and foremost, the most beneficial reason to use FDM 3D printing is the costs involved. FDM printers are very affordable when compared to other printing methods, and the material can be drastically less expensive. While resin printing is continually dropping in price, the size and cost of resin materials are still more expensive than those of FDM printing.

With the expansion of 3D printing over the last few years, the amount of material options have increased rapidly. FDM printing now allows for printing in flexible, nylon, polycarbonate, and carbon fiber blends. There are even much more high-temp materials available that require an enclosure and active heating, though those can still be a bit pricey. There are likely over one hundred types of materials now available, each with their own strength, heat resistance, flexibility, and ease of print – meaning there is something available for almost any application. In fact, Polymaker has over 5 different types of PLA, each with their own specs making them unique to one another. You can likely find the particular material required for your project, so long as your extruder, hotend, and printer are capable of printing them.

This vast assortment of materials is a lot less expansive when working with other forms of 3D printing , especially when talking about a price range that a consumer is able to afford.

Types of FDM 3D Printing



Many of the axes and solutions I describe are for Cartesian machines, though this version of the book is also directed towards CoreXY, since that has become my personal favorite build type. A Cartesian FDM 3D printer has each axis controlled and moved by a motor independently of one another. The X-axis moves independently of the Y.

The majority of Cartesian machines have the build plate moving back and forth in the Y direction, while the hotend is moved left and right in the X direction, with the entire X carriage being moved up in the Z direction. There are actually quite a lot of printers on the market today that are technically Cartesian in their motor movements, though the hotend is moved in the X and Y direction and the build plate is moved up and down. For the particulars in this book, I will likely be referring to these types of printers as CoreXY. This isn't technically correct, but the majority of makers think of Cartesian machines as those where the build plate moves back and forth. This is a little confusing, but the community generally thinks of Cartesian machines as ones with the build plate moving, such as the very popular Ender 3 and Prusa.

A CoreXY machine is where the X and Y axis are moved dependent on one another via the stepper motors. If you move the Hotend in the X direction, both motors will spin, and the same is true for the Y direction. This is why when I refer to CoreXY, I am referring to the bed moving up and down in the Z direction, and the extruder moving in both the X and Y direction. There are some variations with this, but when I say CoreXY in this book, I am referring to the build plate moving up and down and not rattling back and forth.

There are definitely some benefits involved with a CoreXY printer since you should be able to achieve some tighter tolerances, as well as avoid some Z-wobble and thin parts being knocked over from the bed rattling. Your frame should also be able to print much faster due to the lack of the heavy bed being moved back and forth. Many prefer a CoreXY machine, including myself. They weren't as common in the past, but they seem to be growing rapidly in popularity.

A Delta 3D printer looks and functions quite differently than either of these, since the extruder is suspended above the build plate via three arms in a triangular fashion. These machines have their downfalls but can normally print much faster than a Cartesian with the same specs. I have personally used 3 different Delta machines. They are quite fun and can print very well, but they require a taller frame for the same surface area. The larger you want your Delta printer to be in the X and Y direction, the taller the printer has to be. I personally avoid this because I like to use racks to stack my printers on top of one another, and this can be very difficult with tall Delta machines.

You should be able to use all of the remedies in this book on CoreXY and Delta machines, but some of the dimensions described in the directions and firmware will be different.

There are also two types of extruder setups: direct and Bowden; both of which can have a gear ratio or not. A direct extruder is where the stepper motor is feeding filament directly into the hotend, and is attached to the carriage. A Bowden option is where filament is fed from a stepper motor attached to the frame and feeds the filament over a distance to the hotend carriage. The majority of inexpensive printers come in a Bowden setup, including most Creality machines such as the Ender 3. A Bowden extruder will reduce the weight on your carriage which means you can normally print much faster. The problem with Bowden machines is the difficulty involved with retraction, printing unique materials, and printing with fine nozzles. You will also need to ramp up the retraction settings in order to prevent a “hairy” print, which is covered in the “Settings Issues” chapter.

I do cover Bowden setups in this 2022 edition of the book, but in general I am discussing using a direct extruder, since I prefer those. All fixes will work with Bowden machines, but some of the imagery in the photographs may be different. I will cover the importance of your extruder and hotend in the “Upgrades and Purchasing a New Printer” chapter.

Good Practices

Before I get into the specifics of fixing particular failures, I feel it necessary to go over some good practices to maintain your printer in top condition. Since 3D printers are mechanical machines that experience frequent rattling and movement, they require constant maintenance in order to continue performing at their highest level. Look over the “Mandatory Maintenance” chapter as well to make sure you take proper care of your printer.

Keep a clean environment

You will be surprised at just how quickly everything can get out of hand if you do not keep your printer work area clean. Be sure to throw away excess material since it can eventually get in the way of your gears turning properly. Since there are fans blowing on different sections of your printer, having excess material laying around could blow some stringy old material onto your current print.

Keeping your printer clean includes your build plate. This does not need to be cleaned after every print and will be determined by your specific build plate and adhesion method (as described in the “Bed Adhesion” chapter). I like to clean off my build plate every couple of prints to make sure I have a proper first layer, though I have to admit that I have let it go for a few dozen prints in the past. There is no harm in cleaning your build plate regularly.

If you have an air compressor it can help immensely with clearing your printer of debris and dust.

Print replacement parts that are likely to break in advance

You don't want to wait for a printed part on your machine to break and not have a replacement on hand. If you only have one 3D printer, you are going to have to order replacements, when it could have been as easy as printing an extra set when you first got your machine.

The first thing I did when I got a new machine is print a replacement set of printed parts; the files for which are normally provided by the manufacturer. If not, you can likely find them on Thingiverse. I don't do this quite as often since I now have over a dozen 3D printers, meaning I can print replacements after something breaks on a different machine. But if you only have one printer, you won't have this luxury.

Keep replacement parts to the side and hope you will never need them. I have kicked myself plenty of times in the past when I didn't print replacement parts and had only one or two printers that were operational.

Slow your printer down

Many printing issues can be fixed or diagnosed easier if you run your machine a bit slower. It is possible to run into nozzle clog issues by doing this, but in general, you will have a much higher success rate by printing a bit slower.

You can do this by reducing your print speed in your slicer settings and by reducing your machine's acceleration. You can skip directly to "Settings Issues" if you want to see exactly how this is done.

I personally run my machines much slower than the manufacturer advertises and what many makers say they print at. If you are having issues at all, print close to 40mm/s as your top print speed with a stock printer. This may seem slow, but I actually do not run any of my machines faster than 60mm/s, even if they can. I would rather print slowly but correctly than fast and with potential issues. There are of course exceptions to this rule if you have an expensive, well-built machine, such as the new Voron builds.

Save slicer profiles as you go

Every time you make a tweak to your slicer settings, you should save it as a new file and organize it somewhere on your computer. Some slicers make this easier than others. Cura isn't the best for organizing profiles, so it's a good idea to try out other slicers such as PrusaSlicer, IdeaMaker, and others.

Saving profiles is good practice so that you can go back to profiles you know have worked in the past. This can save you an immense amount of time when printing a unique material that you know you have successfully printed in the past. It will also help you to discern whether you are experiencing a slicing issue or if it is mechanical or material related, since your previous settings definitely worked.

You can personalize profiles to specific machines and for each material you are printing with. This is the best way to hone in and perfect your settings for any given filament.

If you don't save each profile, you can still load one from a G-code. You should save your G-codes in a manner that can easily be remembered if the particular slicer settings are needed in the future. Don't just name your G-codes "Print 5" in a generic folder, since you will not be able to recall it easily in the future.

Properly store your filament

Filament, especially nylon mixtures, can absorb and maintain moisture when kept in a humid environment. This is why new spools of filament are always vacuum sealed with desiccants.

If you do not plan on using your filament for some time, you should properly store it with a dehumidifier or vacuum seal it. If you do not, you may start to experience failures on a spool you have used successfully in the past.

Almost all filament should be stored in as close to 0% humidity as possible. I would set my dehumidifier to the lowest setting – 20% - which always worked great. I used to live in a very dry area where humidity rarely got above 30%, so I did not have to deal with this nearly as much as someone living in a very humid area of the world. Now that I have moved to Texas I have to be much more careful in preserving my material.

For further information, watch my video titled “Material Basics – 3D Printing 103” where I go over methods to store and dry your filament, or refer to either the “Material and their Settings” or the “Stripped Filament” chapters.

Always watch the first layer of your print before leaving unattended

Never start a print and just walk away before watching the first layer print. Even on a \$5,000 machine with auto levelling, and a technician with a lot of experience, it would be inadvisable to not watch this first layer print.

Any issue with finding the proper Z-height can lead to a failed print that can damage your machine. If the print starts too close to the build plate, you may have to kiss your build plate and nozzle goodbye. If the print starts too far from the build plate, you are going to have a massive cleanup on your hands.

I would estimate that 75% of the failures I experience can be diagnosed from the first layer. This is the fastest set of failures to diagnose, but if you don't watch the first layer printing, you will be left with quite a lot of headaches. To this day, I watch the first layer print before leaving the machine unattended.

Replace Nozzles

You would be surprised just how many issues and failures can occur from a worn out nozzle. It is difficult for me to dedicate a specific chapter to this, because it can lead to quite a few different issues in your print.

Brass nozzles will degrade quickly over time, and cannot be used with any

abrasive filaments at all. If you have been using a brass nozzle for hundreds of hours of printing, you will be shocked how much your prints improve when swapping to a new one.

I sincerely recommend upgrading from a brass nozzle to a hardened steel one so that you do not have to worry about replacing worn out nozzles as frequently. There are a few options if you want to upgrade to a hardened steel nozzle. I standardized all of my nozzles to “Nozzle X” by E3D, but as of editing this book there is a new option they have called “ObXidian” which I do not have experience with. They advertise it as an upgraded version of their Nozzle X, so I would assume it performs great.

Be sure to only buy nozzles from reputable manufacturers. If you are using an E3D hotend, buy directly from E3D, Filastruder, Matterhackers, or E3D’s verified Amazon store. Inexpensive knockoffs do not have the same tolerances which matter a lot.

If you haven’t replaced or upgraded your nozzle in a long time, don’t even bother checking the diagnosing section in this book, since your nozzle may be the issue.

Dedicate specific machines or hotends to particular materials if possible

If you happen to own a few printers, I suggest designating particular machines or hotends to specific materials. If you dedicate a machine to only print PLA, it will run into less issues than if it prints many different materials. Some issues that come with switching materials include needing to clean out your hotend, requiring different extrusion/build plate temperatures, and potentially needing different types of nozzles. I personally have one machine with a 0.6mm nozzle Hemera on a Volcano hotend that I use for larger, PLA-only prints. This is my work horse printer. I then have a machine with a 0.4mm nozzle for more detailed prints. I also have an enclosed machine with active heating that I only use for higher-temp materials.

I understand that this is not possible for those who only have 1 or 2 printers, but keep in mind that every time you switch to a different type of material you will have different tweaks you need to make.

Don’t get frustrated

3D printing can involve more annoyances than you may have first imagined. It might take you 20 re-prints just to hone in the proper Z-height and to level your build plate, especially if you are not used to the machine. Or you may

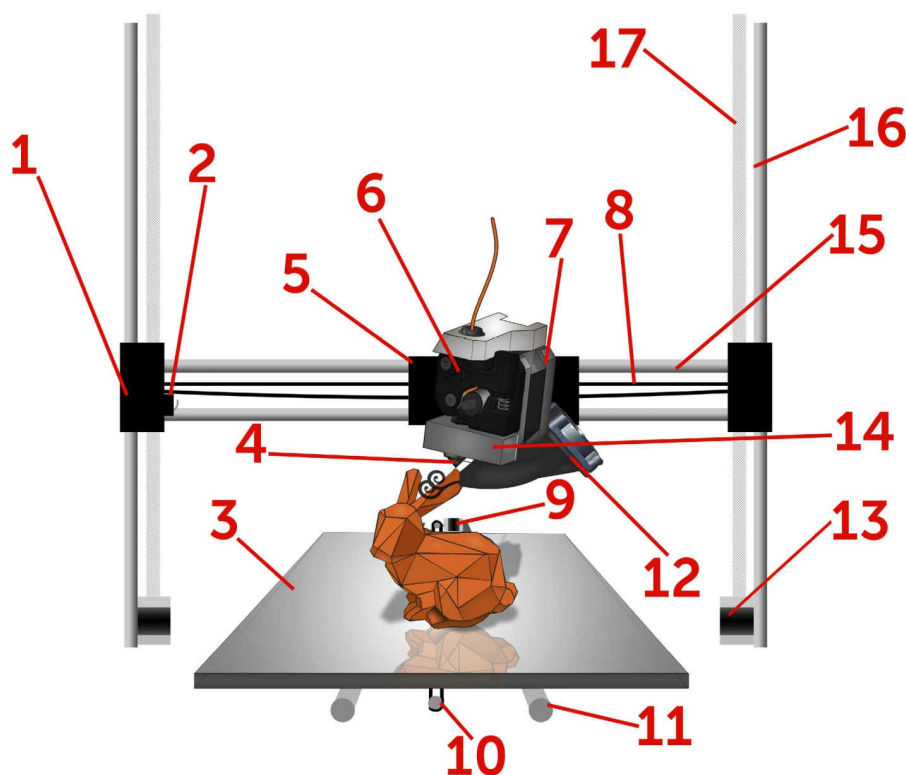
start a 28 hour print that fails 26 hours in.

There are countless ways that 3D printing can leave you frustrated, but it is key to keep calm. Remember that there is a solution to the problem you are having and you just need to properly diagnose it.

Diagram of a 3D Printer

**** Reminder **** All photos in this book, including the diagram below, may have been printed in poor quality for your edition. If you would like high definition color photos, as well as a PDF version, email me at Sean@3DPrintGeneral.com with proof of purchase and I will send them your way within a few business days.

The image below is a printer in a Cartesian setup, where the build plate moves back and forth in the Y direction and the hotend left and right in the X direction; similar to the popular Ender 3 and Prusa printers.



1. Z Carriage:

This connects to both the Z-rod and threaded rod/leadscrew. The leadscrew then turns due to the stepper motor it is attached to, which then moves the x-carriage up and down. On Bowden machines this is often where the extruder is attached.

2. X Endstop: This is what tells the hotend to stop when homing. There is also a Y and Z endstop not shown in this picture, which have the same function (though a Z endstop may be replaced by an auto bed leveler).

3. Build plate: This can be made of glass, PEI, or another material. This is what the prints stick to.

4. Nozzle: Filament is fed through a heated nozzle in order to form your print. These can be found with different diameter holes, and the smaller the hole, the finer the detail. Nozzles range from 0.15mm – 1.2mm in diameter (sometimes thicker with hotends such as the SuperVolcano).

5. X Carriage: This is where the hotend (and printers with direct extruders) attach to. The X carriage is attached to the X rods and belt, which in turn move the hotend in the X direction. This carriage should be very secure and not have any rattling.

6. Extruder: This is how the filament is fed into the nozzle. In this example I am showing a non-g geared direct extruder. A geared extruder will have a gear-ratio allowing for less stress to be placed on the stepper motor, adding a mechanical advantage for more torque, allowing the filament to be fed faster. The extruder includes a tooth drive attached to the stepper motor that pinches the filament against a bearing that freely spins. There are dual drive extruders as well which replace this bearing with another tooth drive. This extruder can also be placed on the Z carriage in a Bowden fashion.

7. Extruder Stepper Motor: The extruder stepper is what turns and feeds filament through the extruder. This would be placed on the Z carriage when on a Bowden setup. This is what you are controlling when you set the E-Steps. When using a geared-extruder, you put less strain on this stepper motor by giving it a mechanical advantage, which would result in less extruder motor skips and a higher E-Step value. It would be smart to place a heat sink on this in order to disperse heat. This added weight when set up in a direct fashion can be one reason people would prefer Bowden.

8. X Carriage Belt: This is what is connected to the X carriage to move it left and right in the X direction via a stepper motor. This belt should be tight/springy to the touch as to reduce Z-wobble.

9. Y Stepper Motor: This stepper motor moves the bed back and forth in the Y direction by controlling the Y carriage belt. This is only present in this fashion on Cartesian machines. Remember on CoreXY setups, there is no “Y stepper Motor”.

10. Y Carriage Belt: This is the belt that is connected to the build plate and is controlled by the Y stepper motor and spins freely attached to a bearing on the other side. Just as with the X carriage belt, this should be tight and springy to the touch.

11. Y Smooth Rods: These rods are what the Y carriage are attached to via bearings and are smooth to the touch. They help to make sure the build plate moves smoothly back and forth without rattling. These rods should be

lubricated with white lithium grease so that the build plate can move without resistance. On particular machines, these can be replaced with a rail system or aluminum extrusion with rollers.

12. Active Cooling Fan: This fan is used to cool prints as layers are being laid down. This is crucial to get clean prints with particular materials, including PLA. Using this fan can lead to decreased layer adhesion on particular materials, so you need to confirm the material you are using before turning it on in your slicer settings.

13. Z Stepper Motor: On some machines there is only one Z stepper motor, but there are dual steppers in this example. This stepper motor turns the Z leadscrew (or thin threaded rod) and moves the X and Z Carriage up and down, via where it is connected to the Z carriage (1 in photo). This is different on CoreXY machines, since those move the build plate up and down instead of the hotend.

14. Heaterblock of Hotend: This is the part of the hotend that gets hot and is attached to the heater. This is attached to the nozzle below it, and the barrel above it (with a heatbreak in between). The barrel should always have a fan blowing on it to prevent heat creep, though one is not shown in this picture.

15. X Smooth Rods: These rods are what the X carriage slides on via bearings and are smooth to the touch. They help to make sure the hotend moves smoothly left and right without rattling. These rods should be lubricated with white lithium grease so that the carriage can move without resistance. These can be replaced with a rail system or aluminum extrusion with rollers on particular machines.

16. Z Smooth Rods: There may only be one of these on your machine, but in the photo above there are two Z smooth rods. These are what your Z carriage is attached to via bearings in order to ensure the Z carriages are moved up and down smoothly without rattling. They should remain lubricated just like the X and Y smooth rods as to ensure there is as little friction with the bearings as possible. These can also be replaced with a rail system or aluminum extrusion with rollers.

17. Z Leadscrew (or threaded rod): These are threaded rods ranging from 5mm-10mm in diameter, with 8mm being the most common. Many machines only have one of these, but I have found that when there are dual leadscrews you get more consistent results. These are turned via the Z stepper motors which then thread into the Z carriages – moving the Z and X carriages up and down. These have essentially the same function for the Z carriages as the belts have for the X and Y carriage. They are threaded rods though because

more weight is placed on these parts, and less frequent moving is required out of the Z direction. In general, the thicker these leadscrews are, the better. Thin 5mm threaded rods can become bent and do not last long on 3D printers.

Electrical Safety

Let me first state my thanks to Timothy over at TH3D since he does quite a lot for the 3D printing community. One area of his expertise resides in upgrades and proper precautions for 3D printers, which many 3D printer manufacturers overlook. He also helped to come up with the content for this chapter.

Please also refer to Thomas Sanladerer's YouTube video "Everything you need to know to make your 3D printer fireproof!" for an even more detailed description of everything in this chapter. He also has a newer video titled "How to make sure your 3D printer won't catch fire!" which I also highly recommend watching.

Always remember, 3D printers are machines and not toys, regardless of how they are advertised. Inside the electronics of your machine you will have high voltage coming from the wall (AC power) and lower voltage that comes out of your power supply (DC power). When working on your machine you should always have the power disconnected and AC cord unplugged from the wall.

While the lower cost 3D printers are great bargains and are helping millions of people get into 3D printing who weren't able to in years past, there are some key wiring issues present on most lower- end brands that should be addressed before using them long term.

Build of your printer

While the actual frame of your printer will likely not be the catalyst for a fire, it can help to contribute to it. This really isn't a problem anymore with the advancements in the industry, but many low-end printer frames used to be made out of wood or acrylic. Wood would obviously make the situation worse, and acrylic also burns quite well.

You can help to mediate any issues when using an all metal frame. Not only are all metal frames preferred for their sturdiness, they also contribute to electrical and fire safety. Luckily, the majority of 3D printers built today have a metal frame.

Even on a metal frame there are plastic printed and injection molded parts that are not flame retardant. As an extra precaution, it is recommended to print in a flame retardant material when making parts for the frame of your machine. Anything that helps to slow down and prevent the potential of a fire spreading is beneficial.

The next part of your printer that needs to be checked is how the wiring is travelling from your board to the parts on your machine. Not only should these wires be organized so they cannot be tugged on during printing, but they should be insulated with a flame retardant material if possible. This can drastically help to prevent any fire that may break out.

You must also consider the operating environment of the 3D printer. Making a wooden box enclosure is likely not a smart idea. Keeping your printer right below curtains is also not a smart idea. Having your printer in an area that is well ventilated and not near flammable objects is the best move you can make in this regard.

Make sure your screw terminal connections are not tinned

The first thing that should be checked on your machine is to ensure that the wires going into the screw terminal connections on your board are not “tinned”. These power connectors are on your board and where your wires from your printer motherboard connect to.

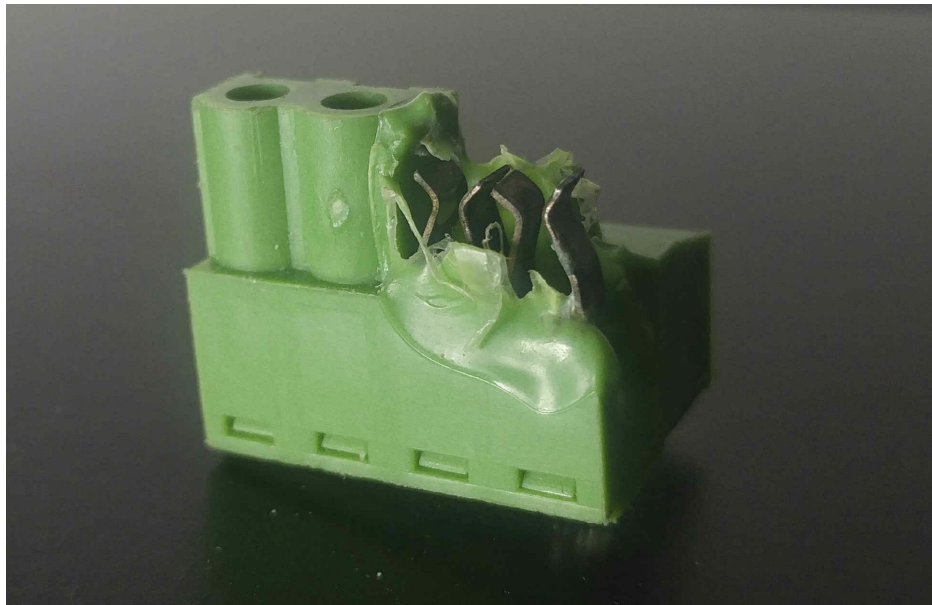
Tinning is the process of dipping the ends of the wires into solder to make sure the wires stay together. This speeds up assembly for the manufacturer and cuts costs down. The issue is that screw terminals should be used with either bare wires or crimp ferrules. Crimp ferrules are ends that you can place on bare wire that compress the strands into a metal end. These are the best case scenario but require the ends and a special crimp to install them. The free alternative is to cut off the tinned ends and strip the wire to expose the bare wire. You can then install the bare wires directly into the screw terminals.

The issue with tinned wires is that when you tighten down the screw terminal it will deform and crack the solder. Then, as current is going through the wire, the solder heats up, expands, and then contracts- leaving a gap inside the terminal. This means that there is little to no contact between the wires and the terminal, which will make a weak connection. Once this happens the electricity will arc inside the terminal, creating a large amount of heat that will melt the terminal. This can result in the printer not working correctly, or even worse, starting a fire.

All wires that are connected to a screw terminal need to be very tight, to the point where you are unable to pull them out with your hands. Check your terminals every so often if you are concerned about this, and if they continually need to be tightened you should not use them. You can get some crimp ferrules, or solder directly to the board bypassing the terminals entirely.

Underpowered connectors

The photo above shows an underpowered connector I used from a RAMPS board. This is extremely common on the power input for RAMPS boards, and can be referred to as a design flaw. The connector that comes with the board is easily overheated and can melt, which is exactly what happened to me on one printer that I built a few years back (as you can see from the photo below). I had no active cooling fan on the board, so if I didn't notice it early, I could have had a real issue on my hands. Luckily this isn't much of an issue in 2022, since most people do not use these RAMPS boards anymore.



Historically, RepRap boards don't use connectors that are rated high enough for the power being run through them. If you are using a RAMPS board, I can't suggest enough that you upgrade the board terminals to a well-made 16A replacement (which can be found at Digikey or Radioshack). This should not be an issue on a pre-built machine, and more for one that you decide to build on your own.

This is actually a very common reason for 3D printer fires. As with other potential fire problems, you should first search the model of your printer and read forums to see if anyone has had problems with underpowered connectors or fires (just as I mentioned the common flaw in the RAMPS board). This will help you save time if you are not super educated with electronics.

You can also manually check if your connector is under powered by finding the part number and looking up what it is rated for. These ratings will include voltage, current, and temperature. The temperature of your enclosed board will be increased, meaning that the connector needs to work in these

increased temperatures.

If you are using a standard 40-80w heater at 12V, and you want to make sure it's good for any situation, all connectors for that heater should be rated at least 8amps at 12 volts, which will allow for anything the heater can throw at it.

This is one of the benefits of 24V printers, since it reduces these issues. A 24V heater will run half of the amperage when compared to a 12V heater of the same wattage. Luckily the vast majority of printers made in 2022 are rated at 24V now, which was not common in the past.

Make sure all of your connectors are from reputable manufacturers and are rated high enough for the power you are putting through them. A good design rule is that you should not be using connectors that are rated less than 120% of the maximum amperage you intend to put through them.

Build plate wiring

Just as with your phone charging cable, the wires coming out of your build plate will be damaged over time (especially on Cartesian machines where the bed is consistently rattling). You can get to the point where the build plate is only being heated by a few strands of wiring that is left, and with the current still being the same across those strands, they begin to heat up more and more. Severe fraying caused by repetitive motion over long periods can create a real potential for a fire.

You can help prevent wires being damaged by creating a strain relief for all wiring harnesses that experience movement during the print. This will allow the stress on the wires to be spread out over a distance and prevent a specific section from having an increased potential of being worn down. Something like flexible conduit or spiral wrap will work great. Drag chains are the best option, but they are a bit more difficult to install.

Finally, you can prevent wire damage by using a good silicon flexible insulation for the wiring. You can purchase new wiring for your build plate and then rewire it all with this flexible insulation as to help prevent this fraying from ever occurring.

Heated build plate MOSFET

A MOSFET is essentially a digital switch that turns power on and off. Most printer build plates will use them on their board to turn heaters on and off.

The issue is that they need to be designed properly for the current you are trying to drive. A lot of times boards will include an under powered MOSFET that is not rated for the frequency or amperage it needs to drive, or are very close to the limitation, without engineering any sort of safety factor. This can cause a thermal runaway. This is particularly dangerous when attempting to print in high temperature or poorly ventilated areas.

These MOSFETS can get hot when pushing more power than they are rated for, and when hot they are also working outside their designed temperature range. One of the best ways to help reduce this issue is to add a heatsink to your MOSFET and to add a cooling fan to your electronics. Keeping the temperature down on your MOSFET will drastically reduce the chances of a fire breaking out.

Technically, the MOSFET should not be so hot that this becomes an issue to begin with. Thermal issues with the MOSFET would indicate that the board isn't using the correctly rated MOSFET. A heatsink and a fan should help to resolve this issue, but the manufacturer should have been using a different MOSFET to begin with. You can de-solder and re-solder a new MOSFET rated for your build plate if this is the case with your printer, but since it isn't an easy process I only suggest doing this if you are comfortable with electronics. This is why a heatsink and a fan is normally the easiest solution.

Some build plates on larger printers work on 115V or higher. It is important to understand the potential dangers of working with AC voltages and are experienced working on 115V or higher before touching this equipment. I don't suggest this for people not experienced with electrical issues.

Thermal protection in firmware

Marlin firmware, especially new versions, have thermal protections built in. This means it can detect if temperatures are outside of the expected ranges and will automatically turn your printer off. These can be found in the configuration.h section of your firmware and should be enabled.

This is why I suggest watching Thomas Sanladerer's video titled "How to make sure your 3D printer won't catch fire!" since it will tell you exactly how you can test if your printer has thermal runaway protection built in.

One example of thermal protection would be with a MOSFET. Your MOSFET can fail either when on or off, and this thermal protection can protect for both. The board checks to make sure the temperature is going down when the MOSFET is turned off, and if the temperature isn't decreasing, the machine will turn everything off as a safety procedure. It also checks the temperature when the MOSFET is turned on, and if it doesn't see any temperature raise, it will cut everything off since your wires could be dislodged.

Too many manufacturers still have thermal protection disabled which is highly inadvisable. While I enjoy the Voxelab Aquila X2 printer as a good Ender 3 V2 alternative, it does not have thermal runaway built in, so I cannot suggest it to people. Creality now makes all of their machines with this important firmware addition. A good manufacturer will have this fixed by 2022, but it does you no harm to test to confirm thermal runaway is enabled on your machine.

Fire extinguishing ball

Finally, the last bit of insurance is a fire extinguishing ball. This is a ball that you can hang above your printer that will explode with fire extinguishing liquid when it reaches a certain temperature.

These balls are only around \$40 each and I personally have an AFO Fire Extinguisher Ball above my printers as a last line of defense. You can take all of the proper precautions but nothing will make you feel quite as safe as using one of these. You can at least rest assured that your entire house will not burn down with these above your printers. Remember that many home insurance providers will not cover a fire caused by your printer, so \$40 is the cheapest insurance you will ever buy, and I recommend it to 100% of people using a 3D printer, especially those who print with their machine unattended.

I also have a regular fire extinguisher in my office just in case, as well as a few packs of WDN throwable fire extinguishers. These don't seem to be available on Amazon anymore, but they weren't very expensive and I feel a bit better having them scattered around my office— plus they look kind of cool. I haven't experienced anything close to a fire with any of my printers, but I would rather have too much precaution than too little.

Mandatory Maintenance

The majority of the tips covered in this section are covered elsewhere for specific issues, but these are all mandatory to do frequently to achieve consistently clean prints. I will perform most of these once every couple of months as precautionary steps to not have to worry about fixing a problem after it occurs.

Most of these steps are also covered in depth in a video I made titled “Mandatory Maintenance for your 3D Printer” on my 3D Print General YouTube channel, though that video is a little dated.

Check there is no gaps or rattling from the bearings

Many machines have bearings that allow the X and Y axis to move along their rods, such as the popular Prusa. These bearings are attached to your carriages normally via printed parts. These are not present on a rail or roller systems such as the Ender 3.

What you will want to do is grab your hotend when the machine is off and try to rattle it around. There should be no movement via this rattling in the bearings. The bearings should grip tight onto the metal rods. If there is any rattling, and you have made sure your hotend is setup properly, this rattling may be coming from gaps in your bearings.

Many machines use plastic bearings that can actually be stretched over time. Bearings should be gripping tight onto your metal rods, and if there is any free play, it can result in ugly prints. I have had to replace plastic bearings on all machines that have them after about 6 months of consistent printing. It is very noticeable after switching to new bearings that the rattling is entirely removed.

If you are using metal bearings, this is very unlikely to be the issue. That said, you will want to make sure these bearings themselves are harnessed tightly to your carriage. This may mean you will need to print new parts or use new zip-ties, since the bearings should be held as tightly as possible to your carriages.

This same exact process should be done for your printer bed, assuming you have a Cartesian machine. In the video I made mentioned in the introduction to this chapter, I had to replace all of the zip ties holding the bearings to my build plate, since they stretched over time.

Luckily printers now shouldn't have bearings held on via zip ties, but this was more common in the past than you would think.

There are many printers on the market that do not use standard bearings, but rather use rollers that move across an aluminum T-slot, such as the very popular Ender 3. These rollers should also be held tight to your aluminum extrusion frame, and if you get rattling, you should grab a wrench and move the nut that holds one of the rollers on. This isn't a normal tightening process, since you can spin these nuts indefinitely. Instead there are sections on the nut that will make everything tighter. Turn this nut until the rollers are held on tightly to their frame. You shouldn't be able to spin a roller without moving the carriage.

Tighten all belts

Other than confirming all harnesses are tight and that there is zero rattling on the extruder and build plate, the next most common reason for Z-wobble is a loose belt. Minor Z-wobble will not be extremely noticeable, so it is important to do check this frequently to keep prints to their proper dimensions.

As explained elsewhere in this book, it is possible to over tighten a belt, but it is pretty difficult to do so on low end machines where the belt is just held together via zip ties. Both the X and Y axis belts should be very springy to the touch with zero droop. There is no real method to measure if your belts are properly tight, you just want to make sure there is no droop and that the belt isn't being stretched. Somewhere in between those two is the ideal range.

If there is any droop in your belt, you will need to tighten. For low end, non-upgraded machines, cut the zip tie that is holding the belt together, grab some pliers, and pull tight as you put on a new zip tie. Make sure the belt is tighter than it was and that the zip tie is pinching everything so that the belt won't slip.

Even better than doing this would be to print a manual way to tighten your belts. There is likely a file on Thingiverse for your specific machine setup, you would just need to search. This is especially true for popular machines such as those made by Creality. In fact, newer Creality machines such as the Ender 3 V2 have knobs to tighten belts built in. If your printer does not have a way to easily tighten belts, this is an upgrade that will save you a ton of time and headaches. Sometimes this involves disassembling your belt to install, but it is well worth it and you will only have to do it once.

Be careful when adding one of these, since you will now be able to over tighten, which I had mentioned is difficult to do without this. Just turn the knob until the belt is very springy to the touch. There is no real scientific way to do this, you just want to make sure there is zero droop whatsoever.

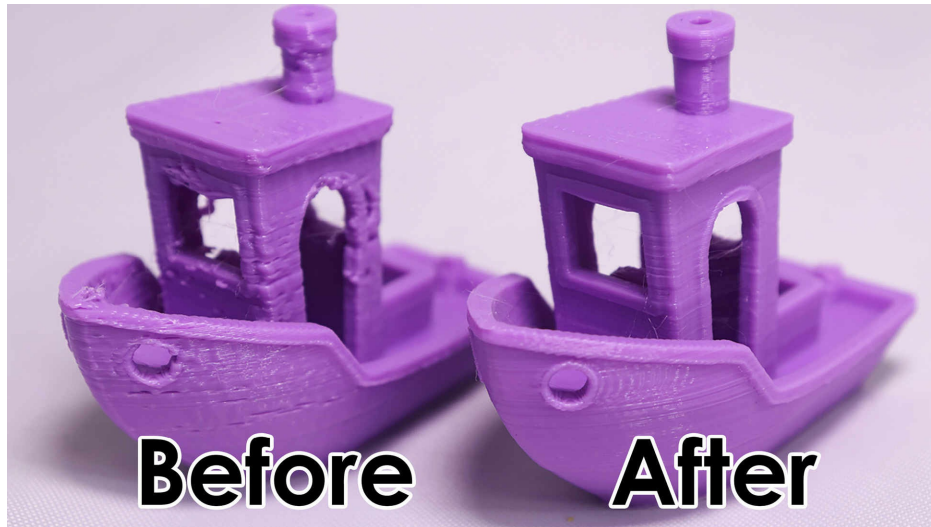
Clean up wiring

Just because you aren't experiencing an issue doesn't mean you shouldn't keep your wiring neat. Tangled wires from disorganization can lead to potential layer shifts or ripped out wires. If your barrel cooling fan's wires are ripped out mid print, you will get a nozzle clog – one that could have been prevented if you made sure your wiring was organized.

Make sure everything is tucked away and that there is no chance for any wire to get snagged mid print.

Replace nozzle (especially if brass)

As mentioned elsewhere, hardened steel nozzles do not need to be replaced nearly as often as brass ones, but replacing to a new nozzle can never hurt. I would replace brass nozzles just about every month or two to ensure that I don't run into any issues in the future, but I have since upgraded to only using hardened steel nozzles. Below is a photo of a print with the same settings, the only difference is using a new nozzle.



Though a little pricey, my favorite nozzle right now is the Nozzle X made for E3D hotends. This not only is abrasive resistant, it also has a smooth coating to allow for material to flow as easily as possible. E3D now has their new ObXidian nozzle, which is an upgrade to the Nozzle X, though I have not tried it as of editing this book. As discussed in the “Over/Under Extrusion” chapter, even hardened steel nozzles can be worn out over time leading to very ugly prints even with all the proper slicer settings, though they will last long enough to more than justify their pricing.

I can't tell you how many times I have gone crazy trying to figure out why my prints were coming out ugly only to replace the nozzle and have my problems disappear.

Refer to my video titled “The Importance of Replacing Nozzles” on my YouTube channel for further information and comparison images of before and after.

Check E-steps

This is the exact same process covered in the Over/Under Extrusion chapter, but you shouldn't wait until you see a problem before checking. Once every month or so I will check my E-steps to make sure they are on point, and will tweak if necessary. This isn't as needed on well-made extruders, but should be done when using the stock plastic ones that come on most Creality style printers. I don't wait until I see an under or over extruded part since I would rather prevent the problem before it happens.

For most under and over extrusion issues, you will want to check and calibrate your E-steps. To do this is actually quite simple and is also explained in the "Over and Under Extrusion" chapter.

You will want to start off by measuring out 100mm of filament. You can actually measure out even more for a more precise readout – you will just have to account for that in the calculations below. I prefer to use White PLA because it is the easiest to write on, has a low printing temperature, and is cheapest - though you could use any material you have at your disposal. I actually now measure out 200mm, since the more you measure and extrude the more precise you can be, but I will be using 100 for the example and calculations below.

You can do this in whatever method is easiest for you. I found it easiest to measure this 100mm when the filament is already fed into the extruder. You can also do this on a desk before feeding, but 3.00mm filament, and 1.75 near the end of its spool, are quite hard to keep from rolling back up.

Be as precise as you can by using a fine tip sharpie and holding the material as straight as possible. Use calipers if you have them at your disposal. After heating your hotend, you then want to push the filament down until the lowest dot you made lines up with the top of your extruder, or somewhere else you can easily line up the starting point (because you will need to compare it to where it finishes). If you line this dot up to right where the filament is fed in, you won't be able to see if you over extruded, since your finishing dot will now be inside your hotend/extruder.

The next thing you will want to do is to tell your printer to extrude 100mm. This is done with a simple G-code command in your terminal.

If you normally print via SD card you will need to hook up to a computer for this. If you print via Octoprint or a similar online program, you can send the G-code commands from their terminals. I personally have most of my printers hooked up to Octoprint via a Raspberry Pi, but this likely will not be common for those of you who are new to 3D printing.

If you do not have your printer connected to Octoprint, you will want to download Repetier Host to your computer, unless you have another software you can use when your printer is hardwired. Normally just choosing “auto” for your baud rate will work, but if you have difficulty with your particular printer, just search online for your printer name and “baud rate”. When hooked up to Repetier Host, or whatever program you use to control your machine, and with your hotend hot, you will want to give your machine the command:

G92 E0

This sets your extruder to 0. Next you will want to give either the command:

For 3.00mm Filament: G1 E100 F30

or

For 1.75mm Filament: G1 E100 F60

The “F” in this equation is just referring to the speed, so you don’t need to use those exact numbers, this is just what I use to make sure we don’t get any extruder motor skips and we are feeding exactly what we think we are feeding. You can always go slower or lower with this number, it will just take a longer amount of time to feed your filament. This will tell your extruder to feed 100mm, and is why it was important you lined up your starting dot with either the top of your extruder or something else that is easy for you to compare to.

Once your extruder has finished you will want to mark your filament at the same spot you lined up your original dot (top of extruder in my examples). If your 100mm dot lines up perfectly, then your E-steps are right on - but even 1mm means that your printer is extruding incorrectly by 1%.

After marking where 100mm actually was, you will want to compare it to where you measured 100mm to be at the beginning of this process. If higher on the filament, your printer is over extruding, if lower on the filament, your printer is under extruding.

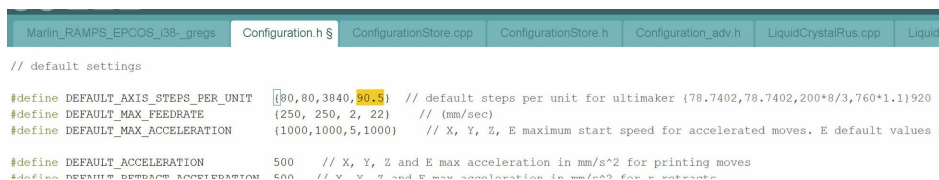
After measuring this difference you will want to write down somewhere how much your extruder actually fed. If your printer over extruded by 2.1mm, you will want to mark down 102.1mm. If it under extruded by 2.1mm, you will want to mark down 97.9mm. You will need this number later on.

The next step in this process is to determine what your current E-steps are. You can do this by either checking the firmware for your machine, by going into the “Motion” section of your LCD screen if available, or just by giving it the command “M503” in your printer terminal. This means you would type “M503” into Octoprint, Repetier, or whatever software you decided to use

that you can send G-code. This “M503” will spit out all of your printer settings and you will need to scroll up a bit to see what your printer has set for its E-steps.

Non-gearbed extruders have E-steps of around 90, while Greg’s wade and other geared extruders can have E-steps of 500 or more. Something like the Titan has a starting point of 420. The Hemera is closer to 400. If you have an extruder from a popular manufacturer, they will list what their standard starting point for E-steps should be. It will help to set your E-steps to this manufacturer recommended number before running your E-step test if you are swapping extruders.

If you are checking in the firmware that you use to flash your machine, you will want to open it up. I actually rarely do this because sending G-code commands is much easier. While in Marlin you will go to the “Configuration.h” tab and scroll all the way down to where it says “DEFAULT_AXIS_STEPS_PER_UNIT”, with e-steps being the 4th and final number (if using one extruder). The X, Y, and Z steps should never be changed and are a calculation based off of the parts you are using.

A screenshot of the Marlin Configuration.h file. The file is open in a text editor with tabs for 'Marlin_RAMPS_EPCOS_138_gregs', 'Configuration.h', 'ConfigurationStore.cpp', 'ConfigurationStore.h', 'Configuration_adv.h', 'LiquidCrystalRus.cpp', and 'LiquidCrystalRus.h'. The 'Configuration.h' tab is active. The code shows default settings for the printer. The line '#define DEFAULT_AXIS_STEPS_PER_UNIT {80,80,3840,90.5}' is highlighted, with '90.5' in yellow. Other settings include DEFAULT_MAX_FEEDRATE, DEFAULT_MAX_ACCELERATION, and DEFAULT_ACCELERATION.

```
// default settings

#define DEFAULT_AXIS_STEPS_PER_UNIT {80,80,3840,90.5} // default steps per unit for ultimaker (78.7402,78.7402,200*8/3,760*1.1)920
#define DEFAULT_MAX_FEEDRATE {250, 250, 2, 22} // (mm/sec)
#define DEFAULT_MAX_ACCELERATION {1000,1000,5,1000} // X, Y, Z, E maximum start speed for accelerated moves. E default values s

#define DEFAULT_ACCELERATION 500 // X, Y, Z and E max acceleration in mm/s^2 for printing moves
#define DEFAULT_RETRACT_ACCELERATION 500 // X, Y, Z and E max acceleration in mm/s^2 for retract
```

The easier method is to just type “M503” into your terminal to be given a readout of what your current E-steps are.

After running your 100mm feed out test, you then take your current E-steps number that you found via your firmware or via the “M503” command and multiply it by 100 (or the amount you were attempting to extrude if you decided to test 200mm instead). You will then divide this new number by the number you wrote down earlier.

For example, if your current E-steps are 90.5 as shown in Marlin above, you will multiply it by 100 to get 9050. We will then divide 9050 by how much you actually extruded earlier. So if you extruded 102.1mm (over extrusion), you will take 9050 and divide it by 102.1 to get 88.64.

$$90.5 \times 100 = 9050$$

$$9050 \div 102.1 = 88.64$$

88.64 in this above example would be your new E-steps. As you can tell it is lower than it was before, because in this example you were correcting for over extrusion.

You will now set your E-steps. You can do this through your terminal,

EEPROM, or by flashing your firmware. If you are going to do this through your terminal, which is my preferred method, you will want to give the M92 command, by typing **"M92 E88.64"**. You will then want to type **"M500"** in order to save these settings. Without typing "M500", the E-steps will be reset when turning off your machine. Remember to have an SD card in your machine when doing this "M500", since some printers require it to save properly.

While you can set this number on LCD screens under the "Motion" section on some printers, it will only save permanently if you have the option to save your settings after doing so, just as with typing "M500" in the example above. Otherwise your E-steps will reset once you turn your machine off. Unfortunately many printers made today actually do not have this options, meaning you will need to do it via G-code commands or flashing firmware.

Thomas Sanladerer has a great older tutorial video going over all of this on his channel which you can find by searching "calibrating your extruder" on YouTube. Thomas really knows his stuff and I suggest to everyone that they follow what he does, since he is my personal favorite YouTuber on 3D printing information. That said, there are newer videos on YouTube which will help walk you through this with particular machines, in case you are having problems.

Cold Pull

When switching to a material that prints at a lower temperature than your previous filament you will likely want to do a cold pull. Cold pulls are also very beneficial to do as regular maintenance on your machine regardless as to prevent oxidized material being stuck in your hotend.

I personally like to perform cold pulls with either a cleaning filament or Nylon mix, but you can perform them with the material you are trying to clear out. My favorite material to do this with is actually Nylon 910 by taulman3D. Not only is that material great to print with, it seems to work even better than cleaning filament I have used in the past for removing the oxidized material in the hotend. That said, it is not needed.

What you will want to do is heat the hotend to the temperature of the material you are using to do the cold pull (250°C for Nylon 910). Push the filament through for an inch, or as much is required for you to no longer see the previous material coming out the nozzle.

Then quickly set your hotend to 130°C - 150°C (I normally do 130°C). You don't want to leave the material sitting in the hotend for a long period of time because it can oxidize itself. Once the nozzle cools to this newly set temperature you will want to pull out the filament. This can be difficult if there is a lot of built up residue material, but it normally doesn't require too much effort.

Once you pull you should see excess burnt or colored material on the filament you just cold pulled. Repeat this process until you no longer see this residue.

This is the best way I know of, other than purchasing a new hotend/heater block, to get rid of the excess and oxidized material.

Clean your build plate

Cleaning your build plate frequently will help to prevent bed adhesion issues. Don't wait until your print won't stick before you clean your bed – proper precaution is important.

Depending on your build plate and what you were using for adhesion will determine how you clean, but I suggest to everyone that they clean their plate after every 5 prints or so. This is especially important when working with PEI beds, since they will gradually lose their adhesion as they become dirty.

In fact, PEI needs to be cleaned very frequently since they will lose their adhesion rapidly if not. When using PEI, only use isopropyl alcohol and do not use acetone, since acetone will ruin the surface.

Lubricate rods and leadscrews

Grab a paper towel and clean off all of your metal rods. Then get some white lithium grease, or your preferred grease or lubricant, and rub it over both your smooth rods and threaded rods. While the bearings you use are often advertised as “self-lubricating”, they don’t last forever.

After adding some grease, move all of your axis to all positions. This will help spread the grease out and you should notice a distinct difference in moving your carriages around.

When using a printer with rollers, such as the Ender 3, you should only have to lubricate the Z leadscrew.

Tighten all screws and bolts on your machine

Frequent rattling of your machine can get screws and bolts loosened on your machine. If you haven't checked in a while, you will be surprised just how many aren't tight anymore.

When I say check every screw, I mean every single one. The ones holding your frame together, the ones holding your build plate, the ones holding your extruder – all of them. This isn't as needed when using T-slot frames, since those seem to hold pretty well, but regular threaded screws do get loose over time. The last time I did this on a standard I3 frame I had waited about 2 months. I found over 5 screws that had become entirely loose. Consider adding non-permanent thread locker to your screw threads to reduce the impact of vibrations over time.

These loose screws can lead to Z-wobble or entirely failed prints. Take the precautionary steps to prevent this from ever happening.

Update your firmware

If you have had your printer for a while, it is likely that the manufacturer has released updated firmware. This firmware can fix bugs or just allow for a better overall experience, so I recommend searching your particular printer online to see if new firmware has been released.

Flashing firmware on older machines can be a bit complicated, and may even require you to bootload your board to add Marlin. Newer boards, such as those on the Ender 3 V2, are much easier to flash. Something like that Ender 3 V2 just requires you to download the new firmware from Creality's website, then transfer over the .BIN file over to your SD card. Power off your printer, put in the SD card, and then turn the printer on. Your printer will now flash with the updated firmware – much simpler than having to bootload.

Remember though if you flash your firmware, any unique numbers you set will be reset to factory. This includes the all-important E-steps. So if you have changed anything, make sure you mark the numbers down. Most settings can be viewed by sending your printer the “M503” command, either hardwired through Repetier or remotely through Octoprint.

Material Science

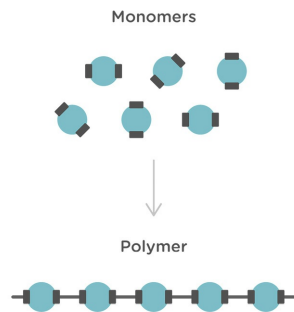
Written by Nicolas Tokotuu, Product Manager at Polymaker

I would like to express my special thanks to the Polymaker team for helping in writing this chapter. Polymaker is an innovative company where research and innovation are the core of the team and company growth. I joined Polymaker in 2016 as a 3D printing engineer and kept growing my polymer science knowledge to the point where I can start sharing with the 3D printing community.

Polymers

In this chapter we will walk through the common issues and challenges encountered in 3D printing with a material science approach. The idea behind the chapter is to provide more scientific knowledge to common issues in order to easily overcome them. Understanding this chapter can help to prevent the need to reference elsewhere in this book.

To begin, it is important to understand what material is being used in 3D printing: Polymers.

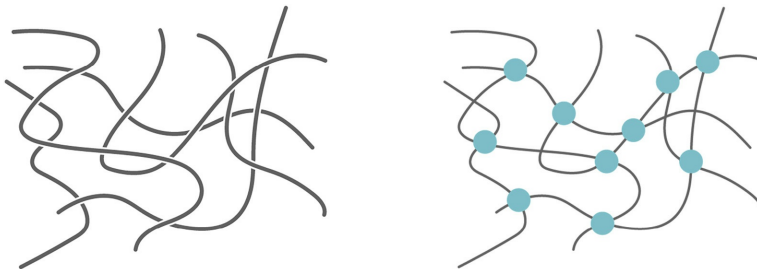


Polymers are large molecules, or “macromolecules”, formed by large numbers of repeating units known as “monomers” in the polymerization process. The polymerization process bonds the monomer molecules together in a chemical reaction, forming the backbone of the polymer.

The type of polymers produced can vary depending on the chemistry and composition of monomer compounds that construct them. The links created between the monomers will be defined as covalent bonds.

Polymers can be divided into 2 families: thermosets and thermoplastics.

Thermosets are polymers that are irreversibly cured from a soft solid or viscous liquid pre-polymer into a solid polymer. The curing process is also known as cross-linking, which proceeds via a chemical reaction that connects all the monomers and pre-polymers to form a network structure. A cured thermoset can no longer be melted and usually is not thermally processable.



Thermoplastics are materials which become soft when heated and hard when cooled. Thermoplastics can be heated, molded and cooled multiple times with

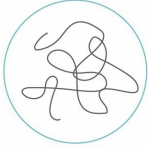
minimal change in their chemistry or mechanical properties. Unlike thermosets where each of the polymer chain is linked to others with a covalent bond, thermoplastics have their polymer chains linked with each other with weaker links which will be defined as non-covalent bonds.

Polymers can also be divided into two main categories depending on their micro-structure:

Amorphous and Semi-Crystalline

One of the ways that different thermoplastics can be identified is through their micro- structure, which can define the properties and behavior of the polymer.

Amorphous



Amorphous

Amorphous polymers are identified for not having a long-range ordering. This means that the polymer chains are randomly oriented.

Generally speaking, clear plastics are often made with amorphous polymers, such as PMMA, PS and PC.

Semi-Crystalline



Semicrystalline

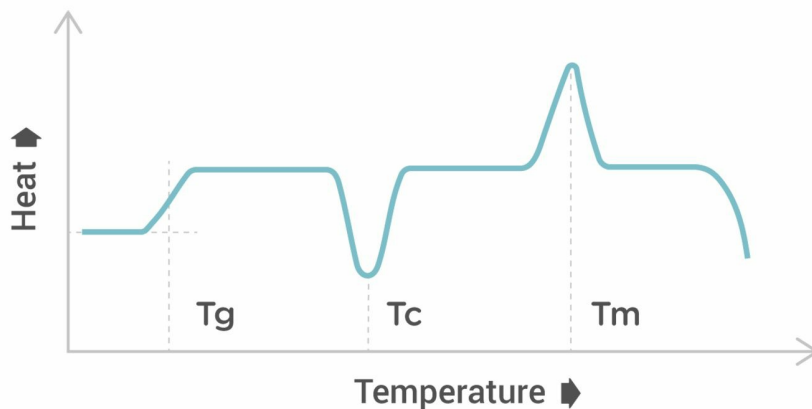
Semi-crystalline polymers are identified for having an ordered structure with structural domains known as “crystals”.

Crystals are an ordered and tightly packed group of polymer chains. Crystalline domains and amorphous domains co-exist in semi-crystalline polymers, thus the “semi”. The proportion of crystallized areas is defined by the degree of crystallinity. A specific characteristic of semi-crystalline polymers is that this degree of crystallinity can highly affect their mechanical and thermal properties.

Now that we have a better understanding of the material structure we will dive into its thermal properties to understand its behavior as a function of the temperature. In order to do that, we first need to define the test which will reveal the thermal properties of a polymer: DSC.

DSC definition

Differential scanning calorimetry (DSC) is a type of thermal analysis in which a specimen is placed within a chamber and the amount of heat required to continually increase the internal temperature of the chamber is measured. This form of analysis is designed to pinpoint the temperatures at which the specimen undergoes certain state transitions e.g. Glass transition, crystallization, and melting, by documenting how a polymer reacts to the gradual heat increase via its level of energy absorption and release.



Glass transition temperature (T_g)

The glass transition temperature can be found in all polymers, it refers to the temperature at which a polymer's physical state transitions from glass (hard & brittle) to rubbery (soft & flexible). The T_g is usually used to highlight the highest working temperature of an amorphous polymer.

Crystallization temperature (T_c)

Crystallization happens between T_g and T_m (melting temperature). It is the process of polymer molecules aligning to form crystals. The crystallization temperature is the point at which the polymers crystallize at the highest speed.

Melting temperature (T_m)

The melting temperature is the point at which the crystalline domains of a semi-crystalline polymer start to melt/deform. Amorphous polymers do not have a defined melting temperature.

Decomposition temperature (T_d)

The decomposition temperature is the temperature at which a material begins to deteriorate, meaning that the backbone of the polymer begins to break down.

Notes about the above graph and definitions

A simple way to understand it is that the heat(energy) injected in the chamber will be used to increase the internal temperature, however if the sample (polymer) inside the chamber absorbs some thermal energy for structural realignment, more heat will be needed to be injected to continuously increase the temperature at a constant rate.

Referring to the graph above, at the beginning a constant amount of heat is applied to the system to increase the temperature at a certain rate. At T_g (glass transition temperature), we can notice that more heat is required to increase the temperature at this same rate, this is because the sample will absorb some thermal energy to break its non-covalent bonds and make the polymers move more freely (resulting in the material becoming soft).

After this phase transition, the sample will have a higher heat capacity, so the system will still require a constant amount of heat to be injected to increase the system temperature at the same rate, but this amount will be higher than before T_g . The energy continuously absorbed by the sample will make the polymer microstructure move more and more freely (excite them). At T_c (crystallization temperature), the polymer chain of the sample will have enough free movement to form crystals. The sample will then release energy (heat) which means that we need to inject less heat to the system to increase its temperature.

The reason is that the crystals structure (a more ordered structure) is coming from a more disordered structure, which will require less energy, thus the release of the extra energy. Once the crystals are formed, no more energy will be released from the sample to the system. However, soon after creating the crystals, at T_m (melting temperature), the polymer chains will continue gaining energy(movement) which will excite them too much and make them break the crystal structure, thus absorbing energy from the system, thus needing to inject more energy in the system to continue increasing the temperature at a constant rate. After breaking all the crystals, the sample will not require any additional energy from the system. This explains the two opposite spikes at T_c and T_m . At T_d (decomposition temperature), the sample will start to decompose, meaning that covalent bonds will start to be broken, the sample will lose its heat capacity and thus less heat will be needed to increase the system temperature.

Warping, Oozing and Overhangs

Now that the thermal transitions and behavior of polymers in function of the temperature are better understood, we can use this knowledge to explain some of the 3D printing phenomena:

Warping, Oozing and Overhangs.

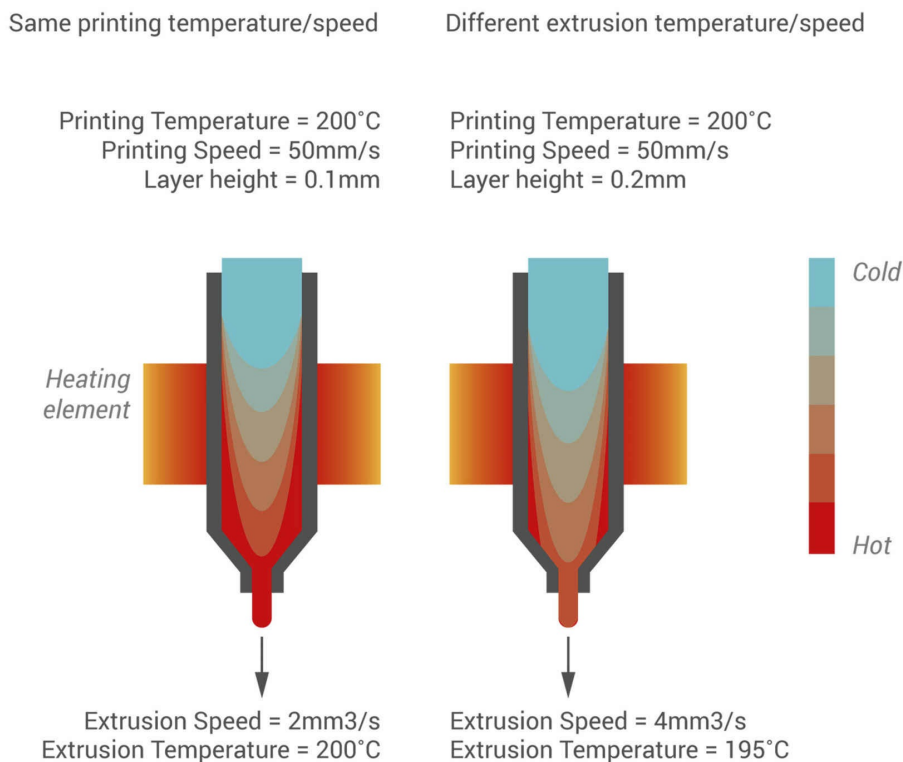
Before jumping into these phenomena, we need to clarify an important point regarding printing speed and printing temperature:

Usually printing temperature is defined as the heat block temperature (in °C) and the printing speed will always define the print head speed when printing (in mm/s).

In this chapter we will refer to more useful factors for us such as the extrusion temperature and the extrusion rate:

Extrusion Temperature: The temperature at which the plastic exits the nozzle (in °C)

Extrusion Rate: The rate at which the plastic is extruded from the nozzle (in mm³/s)



The extrusion temperature can be increased using different factors:

Increase the printing temperature, reduce the printing speed, reduce the layer

height, or increase the nozzle heated chamber length.

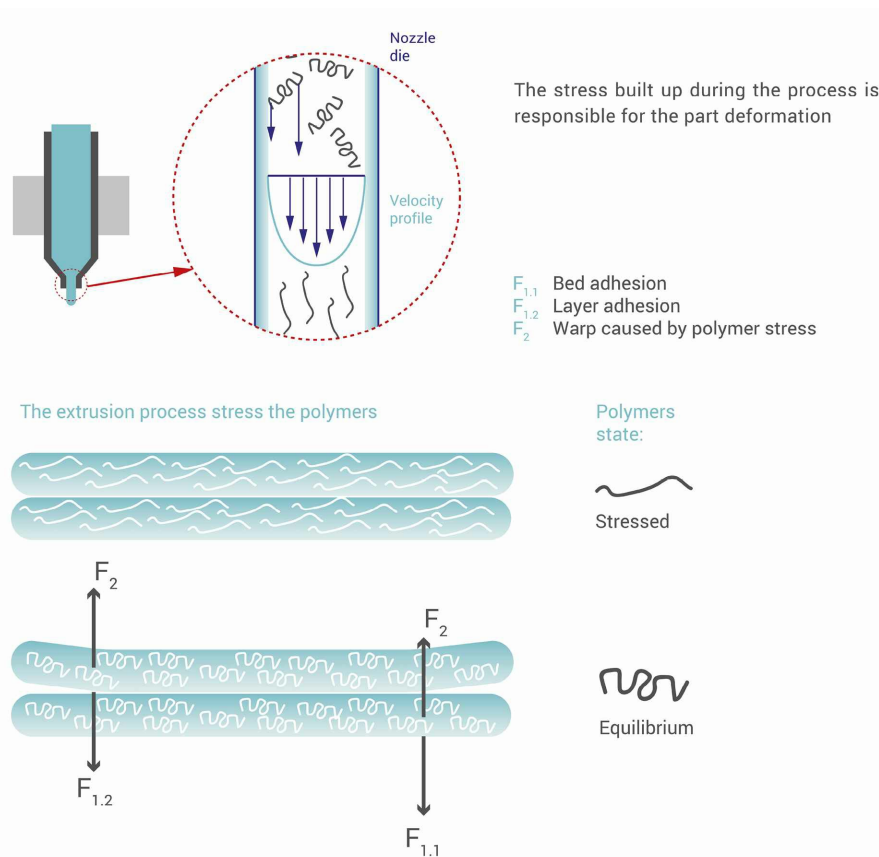
The extrusion rate can be decreased using different factors:

Reduce the printing speed, reduce the layer height, or reduce extrusion thickness.

Warping

In 3D printing, occasionally we will encounter a part that deforms on the printer, curls or lifts up from the bed because of what is known as warping. This is caused by the accumulation of stress created by the 3d printing process.

The origin of the internal stress is still under debate, and depending on your 3D printer configuration, many factors may be contributing to the as-built internal stress. Here is one hypothesis which should be considered for all FDM machines:



During the extrusion process the polymer is forced through a die (small hole/nozzle), and during this step the polymer chains will be stretched to a stress state, then stuck to the build plate or a previous layer of plastic. This stress will slowly be released over time, however if the temperature does not allow the polymer to freely move enough to release the stress, or if the layer is not well stuck to the bed or the build plate, the accumulation of this stress throughout the layers will force the part to macroscopically deform.

Warping and cracking is always representative of this accumulation of stress exceeding the bond between the bed or layer adhesion.

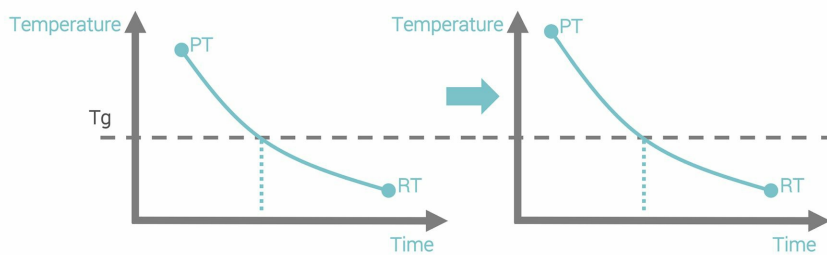
As a result, we have three ways to prevent warping/cracking:

1. Give polymers enough energy to move freely and release their internal stress.

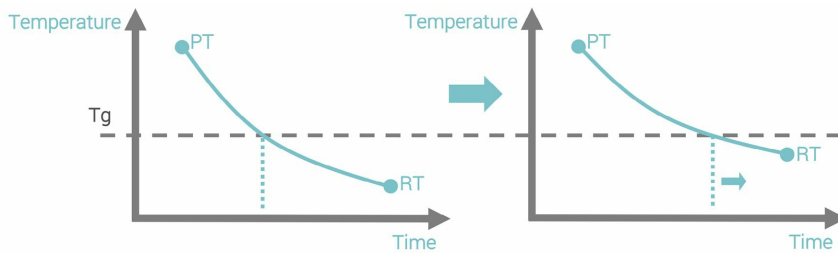
Most of the stress release happens right after the extrusion, indeed the material will be extruded at a high temperature then cooled down below T_g . It is during this time above T_g that the polymer will release most of its internal stress, however if this time is too short, it will not have time to reach equilibrium. Increasing this time period is a way to reduce warping.

This time period can be increased with the following ways:

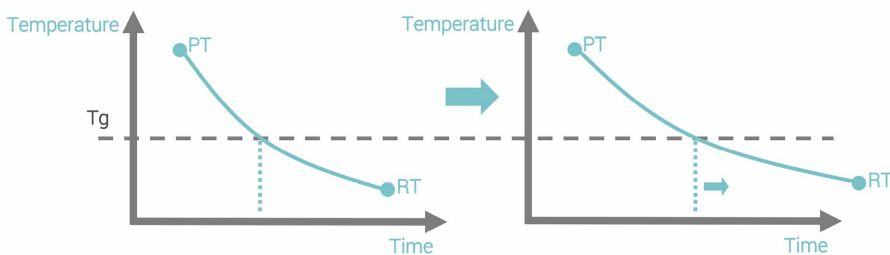
Increasing the extrusion temperature (P_T):



Increasing the room or chamber temperature (R_T):



Decreasing the cooling rate:



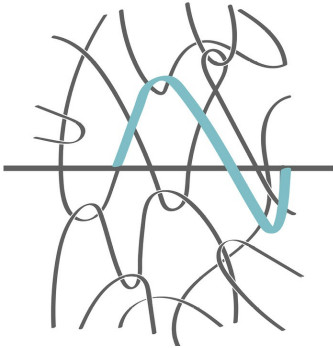
2. Improve bed or layer adhesion

The accumulation of stress will tend to lift up the layer from another layer (delamination) or the bed (warping). However, if the bed/layer adhesion is strong enough to resist the deformation, the polymer will be able to release its stress without deforming the part. The bed adhesion can be improved by using adequate bed surfaces and coating (refer to the “Bed Adhesion”

chapter).

Before talking about how to improve layer adhesion, let us have a look at what layer adhesion is:

Layer adhesion is possible thanks to the entanglement between polymer chains from one layer to another.



This entanglement is possible when both layers are heated up above T_g and both layers have their polymer chains moving freely, and through this movement the chains entangle with each other.

To improve the layer adhesion, we have to increase the number of entanglements between the polymer chains at the layer interface. The number of entanglements can be increased by increasing the time where both layers are in contact with each other with a temperature above T_g . As we can see this is the same solution as number 1. However, an extra factor which can improve the layer adhesion is increasing the contact surface between the layers by increasing the extrusion width.

3. Reduce stress creation

This third solution to solve warping relies on reducing the root cause of warping: internal stress.

As mentioned earlier, the stress is created by forcing the material through a die which will create a velocity curve which will stretch and align the polymer chains. Reducing the stress creation relies on flattening this velocity profile. This velocity profile can be flattened by increasing the nozzle size, reducing the extrusion rate, decreasing material viscosity (by increasing the printing temperature) or coating the internal nozzle surface with low flow resistant surface.

The above explanation of warping can be applied to amorphous and semi-crystalline polymers. However, semi-crystalline polymers face an additional source of stress: crystallization.

Indeed, when printing, the part will undergo crystallization when cooling down creating small crystals which, as ordered structure, take less space and will force the part to shrink. This is why Nylon materials will warp even

though the build plate may only be 45 degrees. If the crystals are formed too quickly, each layer will have small crystals creating a lot of stress per layers and the accumulation of this stress will macroscopically deform the part.

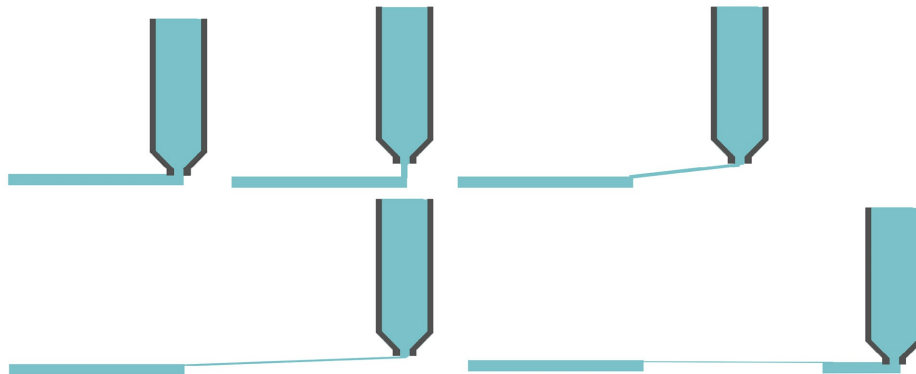
Oozing

In this part we will differentiate two kind of oozing depending on the root cause.

The first root cause is oozing created by the extruded filament being linked with the material inside the nozzle. The extruded filament will then force the material inside the nozzle to stretch out of the nozzle as the nozzle is moving to another location. We will rename this phenomenon as stringing (because of this string created).

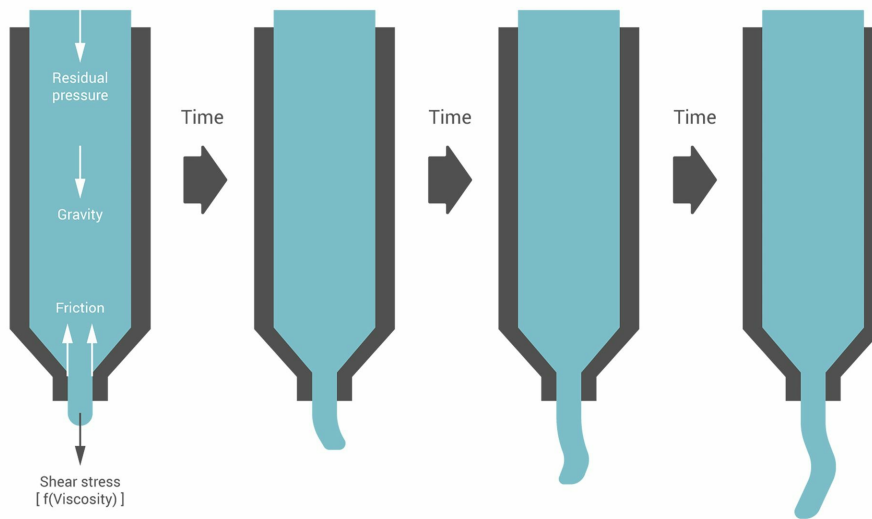
Polymers with a high molecular interaction, or polymers which have absorbed moisture tend to have this issue.

A simple way to solve this stringing issue is to cut the extruded filament from the material in the nozzle by performing a wiping movement with the nozzle before moving the nozzle to another location.



The second root cause is the actual material oozing created by the residual pressure and gravity which will force the material out of the nozzle over time. As mentioned, the above 3 factors will define the amount of material oozing out of the nozzle:

Residual pressure, gravity and time.



In order to reduce oozing, we will need to decrease or counter each of them:

Residual pressure:

Residual pressure is a result of the printer building up pressure within the nozzle to extrude at a certain volumetric speed. This pressure can never be completely discharged from the nozzle over a very short period of time and therefore the material will keep extruding slightly. To decrease the residual pressure, we can increase the retraction settings (distance, speed), increase coasting (using the residual pressure to finish the layer), decrease the extrusion rate (need less pressure to extrude) or increase the printing temperature (need less pressure to extrude).

Gravity:

Gravity will always pull the filament out of the nozzle, and if the gravitational force is stronger than the flow resistance of the plastic against the nozzle's internal surface and shear within the plastic, it will ooze out. Note that the flow resistance between the internal surface of the nozzle and the plastic can be increased by increasing the die L/D ratio (L: length of the die capillary, D: diameter of the nozzle hole). The shear within the plastic can be increased by lowering the temperature of the nozzle (thus the stand-by temperature in several dual extrusion 3D printers).

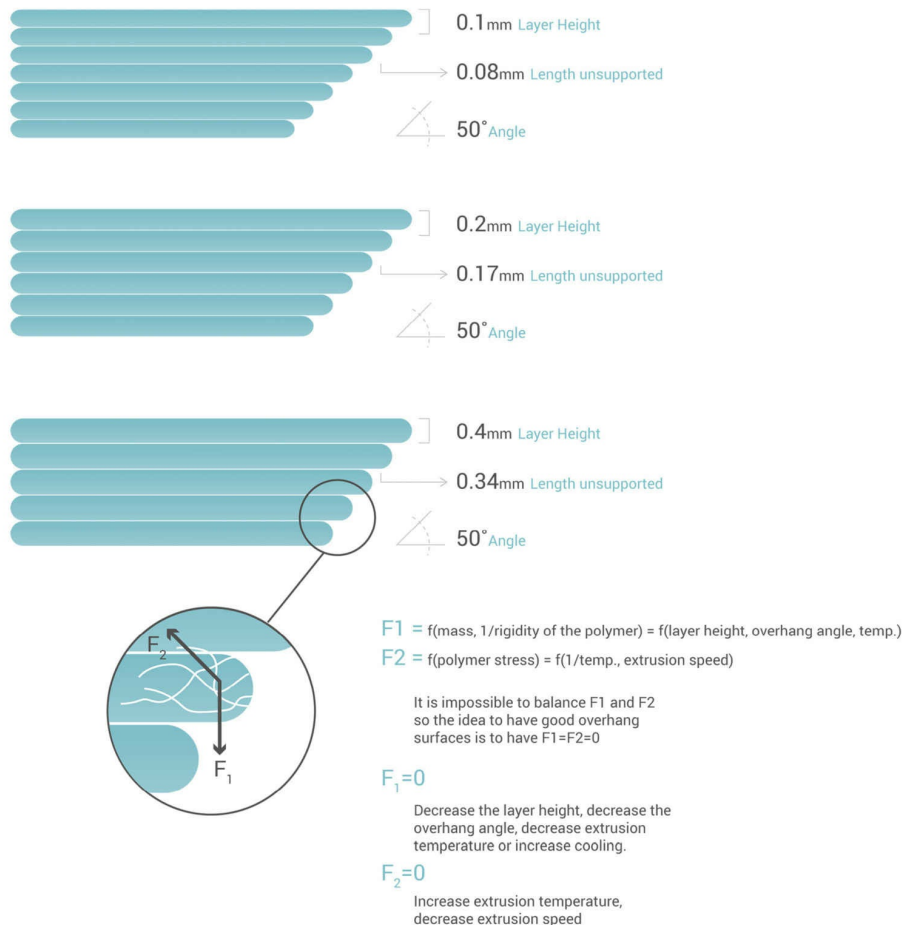
Time:

The amount of material oozing from the nozzle also depends on the amount of time the nozzle is inactive. The greater the duration, the larger amount of material there is. This time can be significantly reduced by having high travel speed, acceleration and reasonably high jerk settings. The material will not have time to ooze out before reaching the other part of the model. Having a

high travel speed and acceleration should not affect ghosting as it would with increasing the print speed and acceleration. However, for dual extrusion printing, this factor cannot really be changed.

Overhangs

Although it is recommended to use support for overhang angle, it usually saves time and material to being able to print high quality overhang surfaces.



The challenge when printing overhang surfaces is the amount of actual unsupported area. As you can see above, the same angle can give different unsupported area depending on the layer height. It can appear that the smaller the unsupported area the better, however the smaller the layer height the less rigid the unsupported area will be. It will always be a balance between rigidity and amount of unsupported area. You can visualize this relationship through one of Sean's videos titled "How to Avoid Needing Support Material" on YouTube.

Different factor can affect the overhang surfaces. As represented on the graphic below two main forces will be applied on the unsupported area: its weight (F_1) and the polymer stress (F_2).

The main factors affecting theses forces is summarized below, however

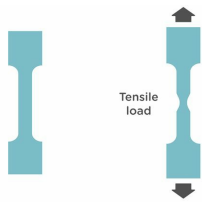
generally speaking the best overhang surfaces will be given with a high layer height (more rigidity), low printing speed (more consistent extrusion) and high extrusion rate (more consistent extrusion).

Mechanical Tests

Before closing this chapter, it can be useful to also learn about the different mechanical and thermal properties which can define a polymer. These 3 tests can determine how “strong” a material is depending on the application you require from your print. CNC Kitchen on YouTube has some great tests of 3D printed parts with the below methods.

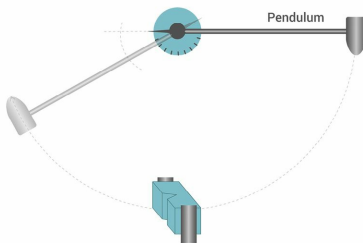
Let us first review the 3 main mechanical tests:

Tensile testing



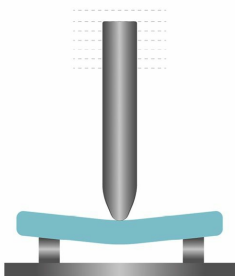
The tensile testing is where a polymer specimen is subjected to tension until it breaks. The test can be used to determine a specimen's tensile strength, Young's modulus, and elongation at break.

Charpy impact test



The Charpy impact test is the process of measuring the amount of energy upon impact that is required to fracture a test specimen. For plastics, the IZOD impact test is more commonly used, but they are similar. This test is conducted by fixing an appropriate polymer specimen in place and releasing a pendulum with a set mass at a set height to collide with the test specimen.

Three-point flexural test

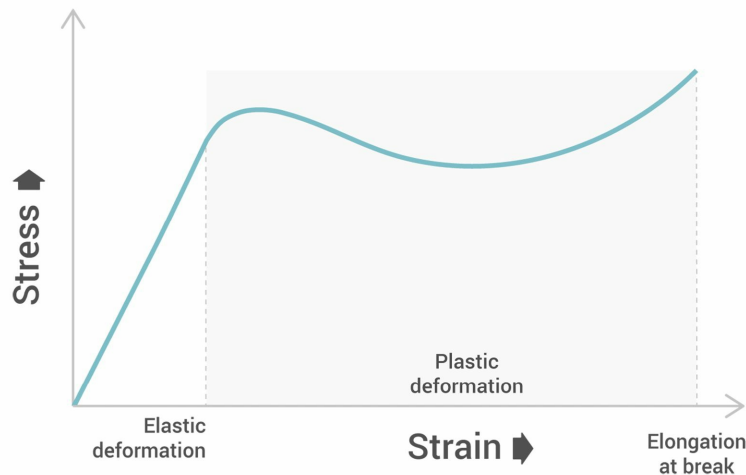


Three-point flexural test is the measurement of a specimen's resistance to deformation under a gradual load. The test samples

are subjected to significant tensile and compressive stresses in their plane in addition to shear stresses. This test can be used to determine the bending strength and bending modulus.

Each of these tests will give important data which will define the material performance:

The tensile strength will give a graph similar to the below one:



Tensile strength:

Tensile strength characterizes the maximum stress required to pull the specimen to the point where it yields or breaks. Tensile strength at yield measures the stress at which a test specimen can withstand without permanent deformation, tensile strength at break measures the stress at which a test specimen breaks, and the ultimate tensile strength is the maximum between both. This allows us to understand the limit of a materials strength and its behavior when under stress. Tensile strength at break is not that common to use since the part has already started necking, usually only yield strength and ultimate tensile strength are of interest.

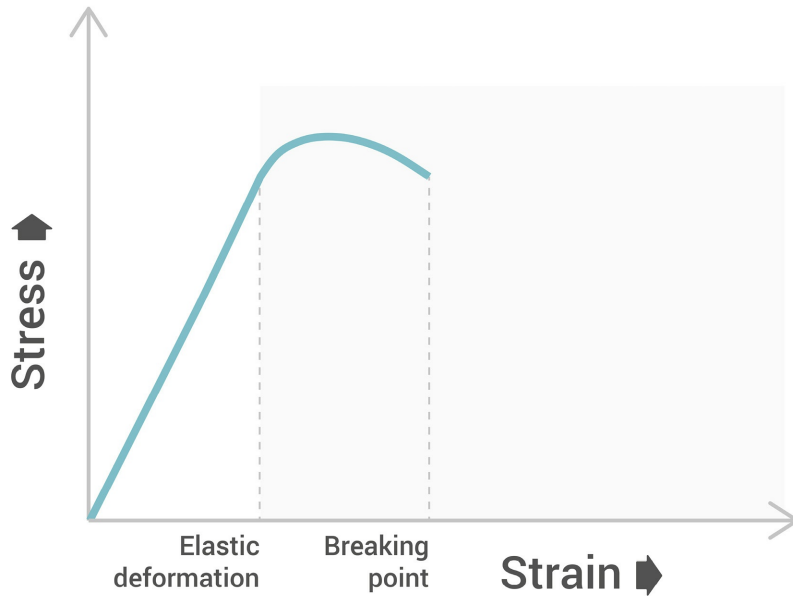
Elongation at break:

Elongation at break measures the deformation ratio between initial length and increased length right before breakage. This allows us to see the amount of stretching a material can endure before breaking.

Young's modulus:

Young's modulus measures the resistance of polymers to deformation under stress along a single axis. The Young's Modulus is a material property that is used to calculate the stiffness of a structure.

The bending strength will give a graph similar to the below one:

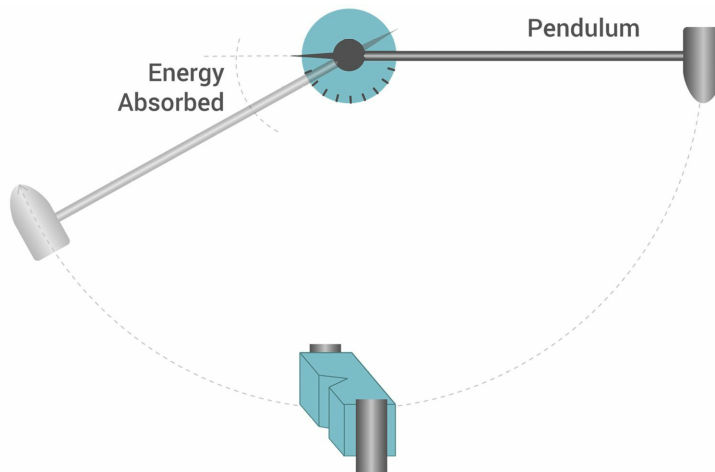


Bending modulus

Bending modulus is a local physical property that is computed as the ratio of stress to strain in flexural deformation. The Bending modulus has similarities to Young's modulus as it tests the polymers ability to resist deformation.

Bending strength

Bending strength represents the highest stress experienced within the material at its point of yield or break.



Charpy impact strength

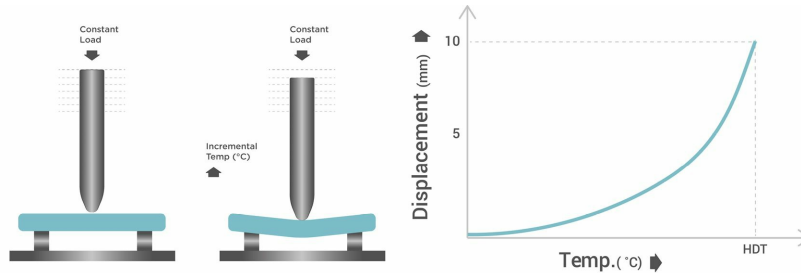
The charpy/IZOD impact tests measures the amount of energy that is required to fracture a material sample under a sudden load/impact. During the test the potential energy of the impact hammer is converted to kinetic energy. A part of this kinetic energy is absorbed by the sample during impact, so the hammer will not swing as further up as in the beginning. The difference

between the initial potential energy and the potential energy of the hammer after the impact is the impact energy. This value is put into relation to the reference area of the sample that broke to calculate the impact strength (kg/m^2).

Thermal Properties

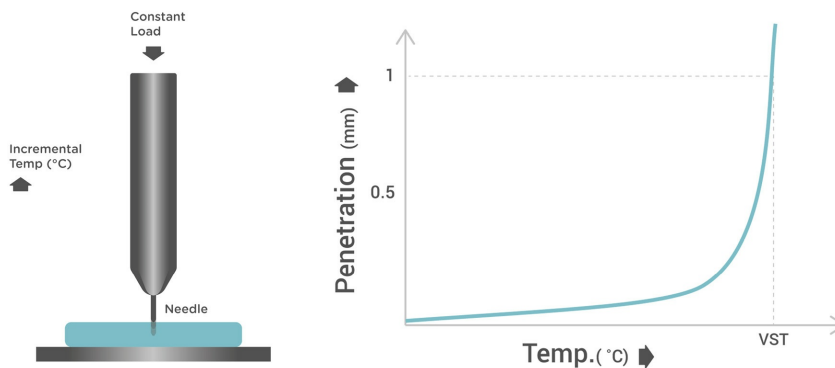
Heat deflection temperature:

Heat deflection temperature is the measure of the temperature at which a polymer undergoes a certain amount of deformation. The test is conducted using a specific load, while steadily increasing the temperature by $2\text{ }^{\circ}\text{C}/\text{min}$ and measuring the temperature once the displacement of the contact sensor of the specimen reaches 10mm.



Vicat softening temperature:

Whilst comparable to HDT, the Vicat softening temperature differs by providing a testing method that simulates the point at which temperature softens the material's physical properties enough for an external object under a set pressure to penetrate the outside surface of the specimen by 1mm.



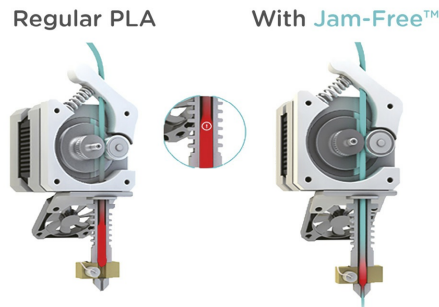
Melt index:

Melt index characterizes the flow behavior of a polymer under a set pressure and temperature. This is achieved by extruding the polymer and measuring the total weight of the extrudate in a set time-period. The more material that extrudes, the increased weight and therefore the lower viscosity.

Polymaker Technologies

Now that we have more material science knowledge it will be easier to understand the different technologies that Polymaker is using in their products to help combat some of these material limitations:

Jam-Free™ Technology:



To understand this technology, let us understand the main root cause of jamming issues:

The print head is divided in two main parts: the hot end and the cold end. The hot end is where the heat block will heat up and melt the filament, the cold end will prevent the heat from the hot end to disperse and damage other components or soften/melt the filament before it needs to be.

However during long prints, dual extrusion prints and simply with a badly design heat sink the heat will climb up to the cold end and soften the filament which can lead to filament expansion thus jam, or the extruder chewing the filament (which we call “heat creep” in the “Nozzle Clogs” chapter).

PLA is the most concerned by this issue because it has a very low Tg (~60°C) so if the temperature raised a little bit over 50°C, it can already create a risk of jam. 2.85mm filament is less concerned by this issue because they are thick enough to stay more rigid than 1.75mm.

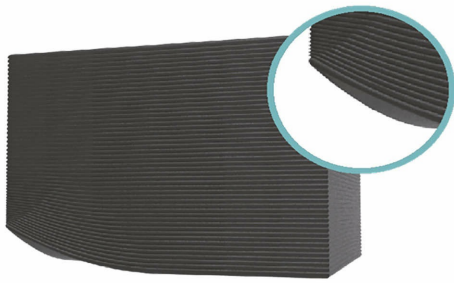
To solve this issue, Polymaker increases the heat resistance temperature of their 1.75mm PLA based product to 140°C.

Since PLA is a semi-crystalline polymer, Polymaker is able to do this by annealing the filament first, which will increase the crystallinity degree of the filament. As we explained earlier, the crystal will start to break at Tm (~150°C for Polymaker PLA) so it provides more heat resistance to the material.

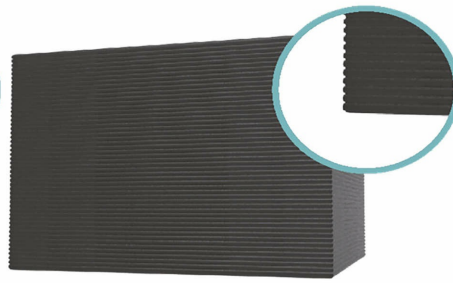
Warp-Free™ Technology:

This technology is used by Polymaker in their PolyMide™ family (Nylon based material). We already learned a lot about warping issues and potential root cause earlier in this chapter. This technology solves one of the root cause of warping issues: Crystalization.

Regular Nylon



With Warp-Free™



Indeed, Nylon is known as challenging to print because of its warping behavior, because when printing, the quick formation of crystals within each layers will create a lot of internal stress - resulting in part deformation.

Polymaker's technology is not only reducing this stress, but it is increasing the mechanical properties of the part. The technology slows down the crystallization rate of the polymer, which prevents it from quickly forming small crystals within each layer as they are printed. Instead, it allows the polymer to slowly build big crystal across layers, since multiple layers have time to be printed before the formation of crystals. These crystals across the layers will also significantly increase the inter layer adhesion. This is also the reason why Polymaker will recommend to anneal the part after the printing process. Annealing ensures the part has reached its highest degree of crystallinity, providing the best thermal and mechanical properties.

You can see how strong this nylon can be by watching Sean's video titled "Nylon Comparison Part 1" on YouTube.

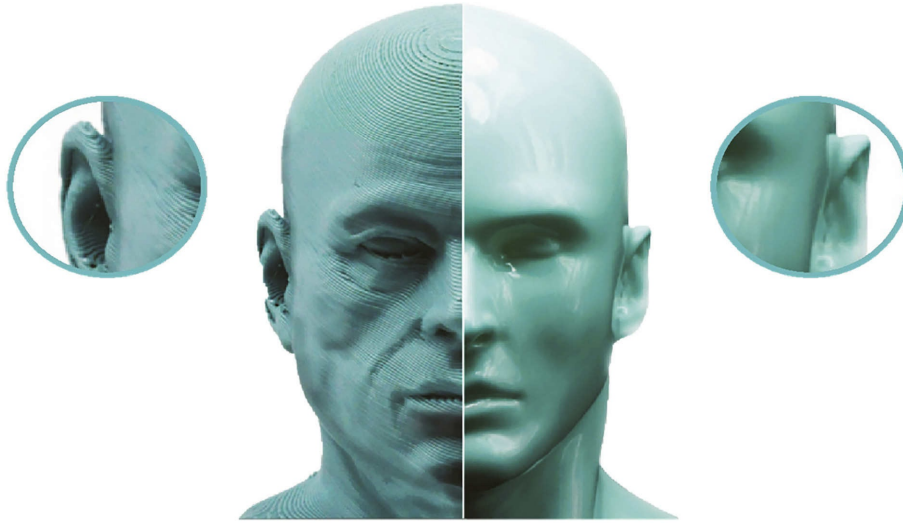
Layer-Free™ Technology:

This technology involves less of polymer science but more a perfect combination of the right material with the right solvent. Polymaker was interested in the smooth results that an acetone bath could give to an ABS print, however they thought that the ABS was too difficult to print, and the acetone could be a dangerous chemical and not safe to use. And there were no actual devices which were designed for the purpose of using this solvent to polish an ABS part (as you can tell in Sean's "Post-Processing" chapter, he used to use a crock pot).

The first challenge for Polymaker was to find a polymer which could be easy to print and also react with a solvent which could be sourced easily and less dangerous than acetone.

Polymaker finally found PVB as the perfect candidate. From there they started to develop specific material formulas PVB based and PolySmooth™

was the results of this development.



PolySmooth™ could be printed with the same settings as PLA and could then be smoothed with alcohol.

The second challenge was to design a device which could safely and reliably polish a PolySmooth™ model using alcohol. The Polysher™ was the result of this device development. The core of the Polysher™ being the nebulizer, the carefully chosen membrane and the specific algorithm developed to find the right frequency for the nebulizer.

Sean has a few videos on the PolySmooth™ going over its possibilities titled “Polysher by Polymaker Review – Smooth your 3D Print”, “Transparent 3D Prints”, and “More Transparent 3D Print Tests”.

Ash-Free™ Technology:

Without Ash-free™
Ash content: 0.5%

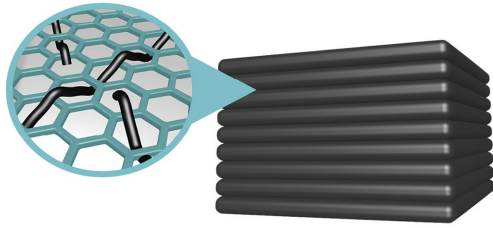


With Ash-free™
Ash content: 0.003%



This technology is used in Polymaker Polycast™. It uses a specific combination of different precisely chosen components to create a material for casting. These components are carefully chosen to burnout without any residues.

Fiber Adhesion™ Technology:

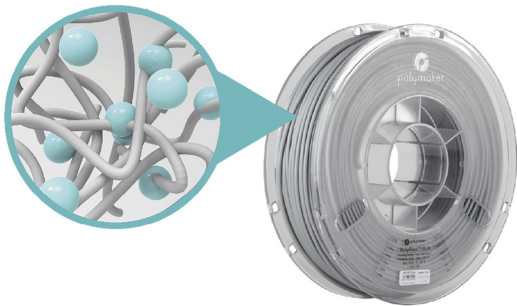


Fiber reinforced materials provide excellent thermal and mechanical properties, however in extrusion based 3D printing, it can negatively affect the layer adhesion. Polymaker believes that the layer adhesion issues come from the fibers not bonding/matching well with the matrix polymer.

After months of development, they successfully optimized the surface chemistry of the fibers to achieve better dispersion and bonding to the matrix.

When implementing this technology to PolyMide™ PA6-CF and PolyMide™ PA6-GF, the layer adhesion was not negatively affected, but actually stronger (+27% for CF, +15% for GF).

Nano-reinforcement Technology:



This technology is used in Polymaker's PolyMax™ family of products. It reinforces the polymer with nano-particles that make it much tougher, while simultaneously keeping similar printing conditions. PolyMax™ PLA, PolyMax™ PETG and PolyMax™ PC are the 3 products from the PolyMax™ family. They all print with the same settings as their regular counterpart: PolyLite™ PLA, PolyLite™ PETG and PolyLite™ PC, however they can achieve up to 5 times the toughness, making them more durable.

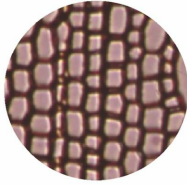
As reviewed earlier in this chapter, the fracture toughness can be well represented by the impact resistance of the material.

Stabilized Foaming™ Technology:

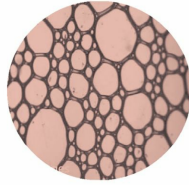
This last technology is one of the earliest developments by Polymaker. After several bad experiences clogging nozzles with printing wood filled filament, they thought about ways which could make a filament look like wood

without actual wood powder in it, since wood powder in the filament could negatively affect the printing process.

Wood



Stabilized Foaming™



Polymaker realized that the main reason for the appearance of wood was its plant cell structure and color. It was easy to copy the color of a certain wood, and the plant cell structure was copied using a foaming agent, creating a similar cell network.

The main challenge was to design and formulate a foam structure which would not be negatively affected by the extrusion process of the 3D printer, thus the “stabilized” in “Stabilized Foaming”, meaning that the foam will remain stable after the printing process. They have finally developed PolyWood™ from this technology.

Materials and their Settings

By this point you are well aware that different manufacturers with different printer setups will require different specific slicer settings. That said, I will review some of the settings that work best for me on both a geared direct drive and geared Bowden machines. These will be good starting points, though the speeds should be turned down a bit when working with non-geared extruders. Much of this chapter is similar to previous editions, though there are some important additions.

While Bowden machines can move faster without experiencing rattling, since the carriage is much lighter without the extruder, they are limited on specific materials due to the distance between the extruder and the hotend. This also means it is more likely to print stringy and have difficulty travelling over small distances without leaving a trail, requiring an increased speed and distance for retraction.

All of the settings below are editable in Cura 4.1 and higher, and likely editable in your slicer as well. Make sure to read the PLA section entirely regardless of what material you are printing with, since there are some overall notes factored within that will be true across all material types.

Also note that speed that I mention, including retraction, can depend on what extruder you are using. Many of these settings are for a Bondtech BMG dual drive extruder, which can handle higher speeds than a non-geared extruder. These are just starting points to which you can tweak later if the quality is not to your liking. Something like the Hemera or similar style extruder/hotend will require lower retraction settings as to not cause a clog. This is because there is an extremely short gap between the heatbreak and the extruder on these setups. Most materials I will have my retraction distance set below 2mm, even down to 1mm, when using those types of extruders.

** This chapter will go over the basics for printing with materials, but does not go into the actual science of what makes up different types of filament. For that, you should read the “Material Science” chapter, contributed by Polymaker. I suggest everyone reads that chapter since it will explain why materials act the way they do.

NOTES ABOUT ALL MATERIALS:

This will be covered in the “Settings Issues” chapter in this book, but first I will cover a few specific slicer settings that will be used regardless of the material. I used to standardize to using triangle infill pattern, though that isn’t always the case now. I recommend everyone watching CNC Kitchen’s

videos, such as “TESTING 3D printed INFILL PATTERNS for their STRENGTH”, where Stefan tests every pattern and compares their pros and cons. I often used “Gyroid” since I like the way it prints, though it does take a longer amount of time. I actually use the basic “Grid” pattern frequently after watching CNC Kitchen’s videos. Keep in mind that some patterns, such as “Hexagon”, may work great, but will require a longer print time since it requires the tool path to cover the same lines multiple times.

I will always have at least 1mm of top layers to prevent pitting and being able to see the inside of the print. I keep acceleration and jerk lower than most other operators because I notice this is one of the most important things to keep low for a clean print, but it does increase the print time required. Both acceleration and jerk can be set higher on a Bowden machine though, due to its decreased carriage weight. Acceleration and Jerk can also be increased when using a CoreXY printer, and can be bumped up even higher on Delta machines.

Make sure you always properly purge the filament when switching between materials in order to reduce the chance of black blobs on your print. Read the “Built up Material on Nozzle” chapter for a further explanation.

Cura also has an experimental feature called “Coasting” that I use for almost every Bowden print. It has been around for a while and most slicers have it, but Cura still has it in this experimental section. You will see throughout this section that Bowden machines come with an increased difficulty reducing stringy and hairy prints. Coasting replaces the last part of an extrusion path with a travel path, meaning the oozed material is used to print the last piece of the extrusion path to reduce stringing. This is great, but if you go too high, you can experience under extruded prints or parts with holes, as covered in the “Missing Layers and Holes in Prints” chapter. For most materials I use the following Coasting settings on Bowden machines:

Coasting Volume: 0.064 (nozzle diameter cubed)

Minimum Volume Before Coasting: 1mm³

Coasting Speed: 90%

Please also note that I now tweak my travel speed settings faster after understanding the “Material Science” chapter. Since travel speed really shouldn’t affect the quality of your print, you should be able to bump this up to the highest you are able to. This not only will save on print time, but should reduce the amount of oozing debris/hair left on your prints. So while I have a travel speed of 150mm/s on all materials down below, I am now playing around with speeds around 200mm/s.

Almost all material types absorb moisture, though some much faster than

others. For PLA, I almost never worry about this, but regardless I still store all of my filaments properly. I have a video on this exact topic called “Material Basics – 3D Printing 103” which I recommend watching. You should always keep the sealed bag that comes with your filament, and I suggest purchasing some desiccants. When storing your filament you should throw a desiccant or two into the bag, and then suck out all of the air before sealing it.

This storage is extremely important for materials like Nylon, but like I mentioned, I do this for all material types to be safe. You could even have a dehumidifier where you keep your filament to help even more to keep filament dry. I currently use an Eibos filament dryer for when materials absorb too much moisture, though any filament dry box will work great, since many companies make them. I personally like the ones that can reach 70°C, since some will max out at 55°C. Both will work, but getting to 70°C will allow for a much faster drying of something like nylon.

This can salvage any material you haven’t used in a while, and is almost mandatory for any long nylon print. Some materials do have a shelf life though, so with those you will never get as high of quality as a brand new spool. When working with hygroscopic materials like nylon or PVA, I will keep the spool in the dryer and let it unspool into my printer for the entirety of the print, since materials like that can absorb too much moisture in just a matter of hours.

If you do not have a filament dry box, you can do a couple of things. The most basic method is to heat your build plate to the temperature that would be recommended in a dry box. 50°C for PLA and PVA. 55°C for TPU. 60°C for ABS and ASA. 65°C for PETG. 70°C for nylon and polycarbonate. You would then put your spool on the heated build plate, put some desiccants on it, and then cover it with a cardboard box. You can use the box that the spool came in. This is not ideal since it doesn’t have any air blowing the moisture out, but it will work. Something like nylon may take over 12 hours though. Keep in mind that your spool holder may get deformed when set to 70°C depending on what it is made out of.

Another method is to use your oven. This is better than the build plate, but many ovens will not go low enough. My oven can get to 70°C but it cannot be set to 50°C. If your oven can be set to the proper temperature, you can put your spool in there for a few hours, likely over 4 hours for something like nylon. Again – your spool holder itself may not be able to handle those temperatures without deforming depending on what it is made out of.

Dry boxes are definitely the best method since they are made exactly for this,

but you don't need it if you only plan on drying out spools from time to time.

PLA

Polylactic acid (PLA) is the most common and easiest to use 3D printing material. While PLA has a very high tensile strength, it has a very low bend-to-break ratio, making it one of the most brittle materials upon impact. That said, companies like Polymaker have come out with a variety of PLA types that go against this conventional thinking. Their PolyMax PLA has a pretty high bend to break ratio with a lower tensile strength, making its properties very similar to that of ABS, in fact even stronger in some regards. They also have a new PLA called PLA Pro, which may become my new favorite type of PLA. That material boasts both a high tensile strength and very high impact resistance, making it very stiff and very durable.

That said, all PLA has a low glass transition temperature meaning that the printed part cannot withstand heat higher than 55°C without deforming, which means parts will deform when left on the dash of a hot car. This also doesn't factor in deformation of thin parts and those under pressure, which will deform at an even lower temperature. These two factors mean that PLA is likely not going to be your choice for mechanical parts that will have to withstand any heat.

While you aren't going to be using PLA as a heat resistant mechanical material, it has numerous other applications. Its ease to print, low shrinkage rate and inexpensive price makes it the perfect choice for models, prototypes for sizing, molding, cosplay, and other fun prints. Then with Polymaker's new options, it can actually be used for mechanical parts as well so long as heat isn't a factor. In fact, PolyMax PLA (and I am sure their new PLA Pro as well) are the preferred material option for those making 3D printed gun parts, which surprised me quite a lot when I learned about it. Because of these increased strength properties, I use these stronger PLA options for all of my mechanical parts unless they require a higher heat resistance. This definitely wasn't true for my past editions of my book, since I used to use PETG a lot.

Basic PLA being a stiff part that holds its dimensions makes it a great choice for anyone wanting to make a negative cast for molding. For these exact reasons I personally use PLA far more than any other material. All of my fan art around my house and most of the examples used in my YouTube videos are made via standard PLA, and I am sure any future mechanical part I need that won't require heat resistance will be in one of those new, strong PLA options.

PLA Slicer Settings:

Print Temperature (180° - 220°C)

This print temperature range is huge and is only this large due to the vast variations among individual manufacturers. Most PLA manufacturers either ship their filament with a processing temperature range of 180°- 205° or 205°-220°, you just have to check with your individual spool. Even then, these ranges are larger than the average material you work with, and you will find that you may be able to achieve a successful print almost anywhere within these ranges. My personal choice is to go to the higher side of the material range. I like to print around 200°C in the materials with the lower range, and around 215°C with PLA that has the higher range on both Bowden and Direct Drive machines - though you can play around a bit and lower if you are experiencing blobs. If you print with a large nozzle diameter and large layer heights, you will need to go hotter than if you were printing with the inverse, since you will need that higher printing temperature for the extrusion temperature to be the same – as covered in the “Material Science” chapter.

Build Plate Temperature (50° - 60°C)

PLA is a bit unique in the fact it can actually print on a bed that is not heated, but it will stick and work best on one that is heated to its glass transition temperature of around 60°C. You can go down to 50°C if you are experiencing “Elephant Foot”, as explained in that chapter.

While working with a 60°C build plate, you can actually just print directly onto a clean glass bed, but it is best to add additional adhesive. My favorite adhesive solution is some Aquanet unscented hair spray since it is easy to add, remove, and keeps the underside of the print clean. This obviously will not be needed with PEI build plates.

When working with a printer that does not have the ability to heat its bed, you will need to go out of your way to make sure the part can stick on the initial layers beyond just hair spray. This is when you will likely want to use blue painters tape or another form mentioned in the Bed Adhesion Chapter, along with including a brim or raft on just about every print. Having a heated bed will definitely reduce your headaches, but it is not mandatory for this material. Luckily almost all printers now come with a heated build plate, unlike those made in the past.

While working with a heated bed, I reduce the amount of times I use a brim to as minimal as possible. This is because most models should stick just fine without warping and removing a brim for PLA is not nearly as easy as it is on other materials. I have definitely regretted using a brim in the past when printing 30 small parts on a single bed – leading to an extra couple of hours

of post-print manual cleanup. If printing a lot of small parts or a lot of small support structures, I would suggest using a raft rather than a brim.

Retraction: (Direct Drive: Distance: 3.5mm – Speed: 35mm/s – Minimum Travel 0.9mm) (Bowden: Distance: 4.8mm – Speed: 55mm/s – Minimum Travel – 0.8mm)

Please note that retraction may need to be tweaked for individual models and extruder types. When using a Hemera I have my retraction set to 1.5mm and 25mm/s, since there is such a small gap between the extruder and the heat break. Some models require retraction to be turned off entirely in order to print properly, but these are good starting points for your average model.

This is the setting that has the most difference between a direct drive and Bowden setup. Due to the pressure built up between the extruder and hotend in the PTFE tubing, Bowden has increased propensity for stringiness and so will require increased retraction settings on every material (some more than others). With a geared direct drive setup, these retraction settings work great on most PLA manufacturers that I have used, with minimal tweaking required for specific models.

Bowden machines need more tweaking than direct drive when it comes to this regard. Some models may be under extruded with these settings and need you to reduce the distance and speed. This is why I often tweak the minimum travel. If you have a part that requires a lot of retraction to reduce stringiness, but there are a couple very small areas on your print that will be under extruded, you can bump this number up so that no retraction occurs over those small sections.

I almost always also keep the COASTING option turned on for Bowden prints, available in Cura 3.3 and newer, mentioned in the ‘Notes about All Materials’ section a few paragraphs earlier.

Speed: (Direct Drive: Speed: 55mm/s - Outer Wall: 27mm/s – Top/Bottom: 30mm/s – Initial Layer: 20mm/s – Travel Speed: 150mm/s) (Bowden: Speed: 60mm/s – Outer Wall: 30mm/s – Top/Bottom: 30mm/s – Initial Layer Speed: 25mm/s – Travel Speed: 150mm/s)

Please note that I normally suggest starting around 45mm/s on non-geared extruders, but since I am running a dual drive geared extruder, I bump this up to the 55mm/s you see above.

This section will get a lot of dispute from other experienced users, and definitely is different than the manufacturer advertised speeds. Please refer to the “Speed Limitations” chapter to see how I go over this a lot further.

Personally, I don't mind a print taking an extra hour or two if it means that the quality is increased and the chance of failing is decreased. Reducing these numbers (as well as jerk and acceleration) from the recommended settings has done nothing but improve my print quality and reduce my failure rate.

PLA is the material I run the fastest due to its ease of printing. A light Bowden allows for fast printing, and if your setup has very little rattling with a geared extruder, you can likely go 65mm/s without any reduction in quality. Due to its rigidity, PLA is an excellent material for fast processing on a Bowden setup because it will not compress much, if at all, in the Teflon tubing.

Anyone saying they can achieve print speeds over 200mm/s is either lying, getting poor quality prints, has a very experimental or expensive machine, or does not understand how acceleration works. As I mention in that "Speed Limitations" chapter, there are exceptions to this rule though, such as the great Voron builds or a well-built Delta. For any print to reach its top speed, enough distance is required and acceleration speeds set high enough to reach these speeds.

If your printer starts rattling when moving in-between parts, reduce your travel speed. If you don't see any ghosting and your parts are coming out extremely clean, you can try bumping speed and acceleration up to see if you get the same results.

These settings are also for when I am printing 0.2mm layer heights on a 0.4mm diameter nozzle. When printing with a very small nozzle at very small layer heights, these speeds will drastically need to be reduced. When I print on a 0.25mm nozzle I reduce everything by about 20%, and I drop my print speed all the way down to 25mm/s when running the extremely small 0.15mm nozzle. An extruder like the Hemera should make it so you can print faster than this with small nozzles.

While you should be able to achieve faster speeds than this when using a large nozzle diameter at large layer heights, you run into the issue of overheating your extruder stepper and getting skipped steps. You can keep speeds as they are if you have a powerful extruder, but it is also likely you will need to reduce these speeds when going over 0.6mm diameter nozzles. When I print with a 0.8mm diameter nozzle at 0.5mm layer heights I also have to decrease my speeds by about 20%.

This is one of the benefits to getting a Volcano or SuperVolcano hotend by E3D. They allow for faster printing with these large nozzles since they allow for a larger melt zone meaning it will get the material viscous even when printing with large nozzles. If you want to print fast with large diameter

nozzle and layer height prints, you will want to invest in one of these Volcano hotends.

I have a video titled “Taking it to the Extreme with the SUPER Volcano” in which I show how these speeds can be much higher when using this hotend. Remember to read that “Speed Limitations” chapter since these speeds can vary drastically depending on your printer setup.

Support Settings: Overhang Angle: 50-55° - Density: 12% - Z Distance: 0.2mm – X/Y Distance: 0.8mm – Enable Support Interface – Interface Thickness: 1mm – Interface Density: 90% - Interface Pattern: Concentric)

These settings are the ones that will require the most additional tweaking for your individual printer, layer height, nozzle diameter, and model. These are also for single nozzle parent support material. If you are using dissolvable support material, you will likely want to increase the density and decrease the Z distance. The 12% density may be too little for specific models, as well as the overhang angle, but since the interface density is so high, the underside of your print should hopefully still be clean. If you have support interface turned off, make sure you increase the density. Your Z distance is entirely dependent on your layer height, since it will always be a multiple of your layer height.

I have found that the support interface works great with PLA with difficult overhangs, it will just require you to be extra careful when removing the support material. Since PLA is so rigid and brittle, a thin part can easily break when trying to remove stubborn support that was not processed with the right settings. With the settings I provided above, it results in a very clean underside with support material that can kind of be “cracked” off in a couple of sections, with a minimal amount of razor cleanup.

There is a video on my YouTube channel titled “Detailed Cura Support Settings” in which I go over this further in depth. I can’t stress enough how much support interface has helped with the quality of my prints ever since it was first introduced in Cura a little over two years ago, and I suggest everyone to check out that video so you understand exactly what is happening, along with examples of my prints.

Your Z distance will need to be tweaked depending on your layer height and nozzle diameter. If your layer height is 0.16mm, it will be impossible for your Z gap to be 0.2mm – Cura will round it up to be 0.32mm. This is because your Z distance will always be a multiple of your layer height, so you can think of it more like “layer gap”, where you tell your slicer how many layers will have no material between your support structures and the underside of your print.

NOTES ABOUT PLA:

PLA is the easiest of the materials you will read about in this section to work with. You can get just as clean results with all of these settings tweaked slightly. As with almost all materials, PLA also can absorb moisture when not stored properly, though not quite as common. If it is extremely brittle to the touch where it breaks without any effort at all, there is also an issue with the spool that will result in poor quality prints. If you can tie your PLA strand into a knot without any snapping, it may have absorbed too much moisture. This does depend on the type of PLA you are using though, since Polymax PLA has a much higher capability to be bent before snapping.

Since PLA is one of the least warping material, you should also always have your active cooling fan on full blast after the initial layer. This makes a drastic difference in surface quality and overhangs achievable.

ABS

Acrylonitrile butadiene styrene (ABS) was the second most used material by myself, as it was always the go-to material for mechanical parts with a low price tag. ABS is what LEGOs are made out of, and its properties make it a great choice for mechanical parts as well as those requiring a high heat resistance, handling a heat deflection temperature up to 95°C (or 205°F). It is also soluble in acetone, allowing you to blend layers together and add an injection molded shine in an acetone vapor bath.

One thing you will hear from just about every experienced 3D printer is the difficulty to print this material without warping and delamination. Small parts may be achievable, but any large model will require a well-built enclosure with ambient air of over 50°C for any hope of success. Even with this environment, a level PEI build plate, and a model with 30 lines of brim, it is difficult to get a non-warped part weighing over 500 grams. This is why I actually have come to love Magigoo – since it works wonders to get ABS to stick and will slide off with ease at room temperature. It is well worth the investment if you are working a lot with ABS or ASA.

You will want to keep your active cooling fan OFF when printing with ABS to help prevent warping. Please watch my video titled “How to Print ABS on an OPEN Printer” for a lot of good tips when printing ABS. While you may not be printing your ABS on an open machine, all of the points I made will still hold true for enclosed machines as well.

ABS Slicer Settings:

Print Temperature (230° - 260°C)

ABS has a higher printing temperature than PLA. Many manufacturers suggest printing between 230-240 degrees, but as Polymaker has taught me, you will have increased layer adhesion and less chances of warping and delamination if you print hotter. I personally now print most ABS closer to the 260 degree mark since it really helps, especially on large parts.

Build Plate Temperature (105°C)

This is one of the reasons it is so difficult to print ABS – bed adhesion. With a high glass transition temperature, you will want to run your print bed at a hot 105°C.

The issue with this will come from having drastically different ambient air temperature. Having cool air surrounding your print will cause those areas to want to shrink before the bottom of your print, leading to warping and delamination.

I have no problems printing small parts in ABS on an open printer, but it would be smart to print anything of size on an enclosed machine. I have been able to reach ambient temperature of over 50°C in an enclosed machine, which works perfectly to prevent warping and delamination. On my actively heated printer, I get temperatures closer to 65°C, which has proven perfect for very large, dense ABS prints. You just need to make sure the electronics used inside the machine can handle those temperatures, and that your board is located outside of this enclosure.

Even when having your print bed at 105°C, warping can still occur. I have standardized to Magigoo for ABS parts, since it really does work wonders for even large ABS parts.

I used an ABS slurry recently, and it wasn't worth it. Use Magigoo if you are worried about your part warping.

Retraction: (Direct Drive: Distance: 1.8mm – Speed: 15mm/s – Minimum Travel 1.2mm) (Bowden: Distance: 3.5mm – Speed: 35mm/s – Minimum Travel – 1.2mm)

For ABS I reduce my retraction settings a bit from PLA. Remember if you are using a Hemera type extruder, you want to reduce these even further. It seems that ABS doesn't have as many issues with "stringiness" or "hairy" prints as other materials, so these numbers don't need to be quite as high. This is because ABS has a lower heat capacity than most of the materials we work with. PLA, PET and elastomers have much higher heat capacities. The lower the heat capacity, the faster the material can cool. This leads to less stringing but also lower interlayer adhesion. ABS is also still fairly rigid so it doesn't have to deal with extreme compression like what would be dealt with when processing a nylon or elastomer on a Bowden setup.

If you are experiencing stringing with these settings, increase accordingly, or use "Coasting" if applicable on a Bowden setup.

As with PLA, each model may require different retraction settings, with even some models having you turn retraction entirely off.

Speed: (Direct Drive: Speed: 35mm/s - Outer Wall: 25mm/s – Top/Bottom: 30mm/s – Initial Layer: 25mm/s – Travel Speed: 150mm/s) (Bowden: Speed: 35mm/s – Outer Wall: 25mm/s – Top/Bottom: 30mm/s – Initial Layer Speed: 25mm/s – Travel Speed: 150mm/s)

As with all materials, speed is also going to be limited by the machine you are using. That said – for ABS particularly I really slow stuff down. While my printer can print ABS much faster than this, printing slow will help dramatically with layer adhesion and reducing the worry of warping and

delamination.

Travel speed can be bumped up if you are using a strong, reliable machine, but you will always be limited based on acceleration and deceleration speeds. This is why a large printer can normally reach higher top speeds than a small one, having a longer period of time to accelerate and decelerate.

These are settings for a 0.4mm nozzle and 0.2mm layer heights. These speeds will have to be decreased with smaller nozzles and layer heights, and can be increased with higher ones. You can increase your print speeds from here, but I still suggest going slow with ABS so that it can release its internal stresses slower and lead to a stronger layer adhesion.

Support Settings: Overhang Angle: 45-50° - Density: 15% - Z Distance: 0.2mm – X/YDistance: 0.8mm – Enable Support Interface – Interface Thickness: 1mm – Interface Density: 90% - Interface Pattern: Concentric)

ABS is another material that I prefer to use the support interface on in order to improve underside quality. From my tests it seems that ABS cannot handle angles that great, especially since we do not use an active cooling fan, so I will often set the overhang angle to 45°.

Since I keep the cooling fan OFF on ABS, it also seems to have more difficulty bridging gaps. Because of this, I bump the density up slightly to 15% just to make sure the support interface lays down cleanly. I personally see no reason to go higher than 90% support interface density on any material, so I keep that the same regardless. Remember that this Z gap will be determined by your layer height, as mentioned in the PLA section.

NOTES ABOUT ABS:

ABS is slowly being used less and less in 3D printing as easier, strong alternatives present themselves and their prices continue to drop. The rate of me getting a failed print while using ABS is much higher than PLA or PETG.

One of the best parts about printing in ABS is the fact it is soluble in acetone, meaning you can smooth the outer sections and make it more watertight, for a more injection molded look and feeling. You can do this via a crock pot on low for 10-20 seconds with your print lifted, you just have to be very careful doing this. For a further explanation, please read the “Post Processing” chapter in this book.

That said – I have actually begun using the next material a lot, and it is very similar to ABS

ASA

Acrylonitrile Styrene Acrylate, or ASA, is extremely similar to ABS, though it has the awesome benefit of being UV resistant as well. While it still requires an enclosure and can warp easily, I have actually begun using ASA very frequently lately for mechanical prints that require heat resistance, and it seems the community has as well.

Many people find ASA easier to print than ABS, but for me it is pretty much the same.

I have personally printed all of my 3D printed firearm projects in ASA since they are mechanically strong and can withstand heat of 95°C. Along with its ability to be UV resistant, I see no reason to use ABS over this, unless your only concern is price, since this does cost about \$5 more per KG.

Every setting on ASA I keep the same as when I print in ABS, except for a couple minor temperature changes. Because of that, I will only be covering those.

ASA has a very similar printing temperature range of ABS, just about 5 degrees less. I personally print ASA at 255°C, though it can successfully print at 240°. I run ASA slow and hot just like with ABS as to reduce chances of delamination and warping. I also use Magigoo with ASA, since it works wonders as well.

The other change is I print with the build plate at 95°C, slightly lower than ABS. Since ASA is so similar to ABS, it is also soluble in acetone, meaning it can be acetone vapor smoothed as well.

If you are thinking about using ABS for your project, I highly recommend checking out ASA. All of my tips in the “How to Print ABS on an OPEN Printer” video will also be applicable to ASA.

PETG

Polyethylene terephthalate with a glycol modification (PETG) is a plastic resin of the polyester family that is used in beverage containers, food packaging, and countless other everyday applications. While it is used for food containers, it is not recommended to use any 3D print for food parts, due to the minor gaps created in-between layer lines where bacteria can grow, as well as tiny metal fragments that could come off of your nozzle.

PETG was becoming the main replacement for ABS, being the go to filament for strong mechanical parts at a low price tag. You can get a 1KG spool of PETG for only a couple of dollars more than PLA, and you can easily print a large piece without warping. That said, with Polymaker's expansion of PLA options that are quite strong, PETG would only be recommended for parts that need a slightly higher glass transition temperature than PLA, but not quite as high as ABS. These new PLA options are actually stronger than PETG, so I prefer Polymax PLA or PLA Pro over PETG for the vast majority of applications now. PETG is also very chemically resistant, so it will be perfect for projects that require that.

I used to use PETG quite often for mechanical parts, but I have kind of standardized to using strong PLA for most parts, or ASA if it needs high heat resistance.

PETG is very susceptible to stringiness due to its high heat capacity, and you can end up under extruding by attempting to rectify it via increasing retraction, so you may need to clean this stringing up post print on specific models.

One other issue with PETG is that it seems to vary quite drastically depending on the manufacturer. I have had PETG that shatters like PLA, and I have used PETG that bends entirely and won't break. That is one other reason it is difficult to recommend for particular projects. If your part needs to be chemically resistant, that is pretty much the only time I now 100% recommend PETG.

PETG Slicer Settings:

Print Temperature (240° - 260°C)

Some say that you can print PETG below 240°C, but I personally have always printed at 250°C successfully (with minor tweaks for small/large nozzles and layer heights). Always check your filament manufacturers recommended settings, since some manufacturers will have a different range than I listed above.

Build Plate Temperature (70°C)

You can actually print PETG without a heated bed, but it isn't simple. When heating to 70°C, and having a fully level bed, PETG sticks great and has no problems with warping. This, like the print temperature, can have a different range depending on the manufacturer.

I have printed parts that take up the entire 300mm x 300mm x 400mm build area of the CR-10 in PETG with absolutely zero problems with warping or delamination, though you will have to be confident on the manufacturer. I personally like Fiberlogy (sold by WolfWorks 3D in the USA), as well as Polymaker for my PETG needs.

If you have difficulty with your particular PETG sticking to your build plate, you can always use Magigoo.

Retraction: (Direct Drive: Distance: 3.5mm – Speed: 35mm/s – Minimum Travel 0.7mm) (Bowden: Distance: 5mm – Speed: 55mm/s – Minimum Travel – 0.7mm)

Many of the settings for PETG are similar to PLA, but I have found a slightly increased retraction to help out immensely to reduce stringiness. As mentioned earlier, PETG seems to ooze out more than other materials, so you will want a bit of an increased retraction over PLA, and you will also want to have Coasting turned on for Bowden prints.

If you see under extrusion after increasing these numbers, reduce these settings and you will just have to clean off the string post-print with a blow dryer or a razor.

Speed: (Direct Drive: Speed: 50mm/s - Outer Wall: 25mm/s – Top/Bottom: 40mm/s – Initial Layer: 25mm/s – Travel Speed: 150mm/s) (Bowden: Speed: 55mm/s – Outer Wall: 30mm/s – Top/Bottom: 30mm/s – Initial Layer Speed: 25mm/s – Travel Speed: 150mm/s)

I keep everything just about the same as I do when printing with PLA. These numbers worked perfectly so I saw no reason to change anything.

Keep in mind, as with all materials, these settings are for a 0.4mm nozzle and 0.2mm layer heights, with speeds needing to be tweaked if you are using something different.

Support Settings: Overhang Angle: 60° - Density: 12% - Z Distance: 0.2mm – X/Y Distance: 0.8mm – Enable Support Interface – Interface Thickness: 1mm – Interface Density: 80% - Interface Pattern: Concentric)

While I keep the actual support settings the same as PLA, the angle which

support is required is higher than PLA, at least when using an active cooling fan. When printing overhang tests, it seems that PETG achieves some of the cleanest overhangs of any material. This means you can go all the way up to 60° without requiring any support material, saving you a ton on material and time required to print lots of models. I have actually achieved 65° without the need of support material, which is quite high for any material.

If your particular PETG requires that you turn the active cooling fan off for improved layer adhesion, such as Polymax PETG, then you will need to have support structures at a lower angle.

With the support interface you may need to get your razor out, but removing support should essentially “crack” off and leave you with a very clean underside.

Every PETG I have ever tested has had a lot of difficulty bridging. So while you can achieve some great angles, you will not be able to bridge large sections without support material.

NOTES ABOUT PETG:

There is a video on my YouTube channel titled “PETG Cura Settings” in which I go over this more in-depth, though that was using a PETG material I do not really prefer any longer. I go over my testing of my favorite PETG in a video titled “Fiberlogy PET-G Review”. It has been a while since that review though, since as mentioned, I do not really use PETG anymore.

I have always used the active cooling fan “on”, but others have said to keep it off for increased layer adhesion. If you are noticing difficulty with layer adhesion, try turning the active fan off. If you are getting some ugly surface quality with the fan off, try turning it on and see if the strength of your part does not decrease.

PETG also seems to have difficulty bridging gaps (when compared to PLA), so you will require support settings turned on for small gaps. Remember I barely use PETG anymore since I have standardized to strong PLA or ASA for mechanical parts.

Flexible Filaments

Flexible filaments come in a wide variety of properties and print settings, so it is just about impossible to give an all-inclusive profile. I have made 2 YouTube videos in which I go over 10 popular flexible titled “3D Printing Flexible Filament Comparison” part 1 and 2 to get a bit more of detailed description of options available.

While there are quite a lot of flexible options out there, my personal favorites

for a lot of applications are Cheetah by NinjaTek or PolyFlex TPU 90-95.

NinjaTek makes NinjaFlex, one of the most flexible options available on the market, but it is extremely difficult to print. Cheetah was created in order to fix this by allowing you to print faster, hence its name.

There are other flexible options that can be printed on a Bowden, such as SainSmart TPU and PolyFlex TPU 95-HF. Essentially the stiffer the material, the easier it is to print on a Bowden setup, though particular filaments like the TPU95-HF is designed to print with a “High Flow”.

Rather than go over each type of flexible filament, since there are just so many, I will go over some notes applicable to most flexible options:

- All flexible filaments print better on a direct extruder vs a Bowden machine, and some flexible materials are entirely impossible to print on a Bowden extruder.
- The general rule of thumb is – the softer and more flexible the material – the harder it is to print, and the more you need a direct geared extruder.
- You will need a geared extruder rated to print flexible filaments in order to print a wide variety of flexibles. The Hemera extruder/hotend setup can actually print flexible filaments at PLA speeds. This is due to the almost nonexistent gap between the extruder and the heaterblock. If you do not have a Hemera type extruder, you will want to print TPU’s much slower than other materials. The larger the gap between your extruder and hotend, the slower you will need to print.
- Slow your print down! Print at speeds about 25% lower than recommended manufacturer settings to start with, and reduce further if needed. For some very soft materials such as 3DXFlex and NinjaFlex I have to print at 25mm/s in order to not experience a failed print. As mentioned though, extruder combinations like the Hemera can actually print soft materials at over 60mm/s.
- If you do print a flexible filament fast on an extruder like the Hemera, you will need to bump up your printing temperature quite a lot. Many flexible filaments advertise a printing temperature for printing slow. The faster you print, the higher you will need to set your printing temperature.
- Almost any flexible filament with a shore hardness of 90A or lower is relatively very difficult to print.
- You may not be able to use smaller than a 0.4mm diameter nozzle, depending on how flexible the material is and what your extruder setup is.
- Flexible options will often string and ooze more than other types of filaments, meaning you will need to hone in your retraction and possible

clean up parts post print.

- Parent support material is extremely hard to remove. Flexible filaments with a large bend-to-break ratio have high layer adhesion, making parent support material very hard to remove cleanly. Because of this, I do not use support interface. You should always factor in the difficulty to remove support material when designing your flexible part. Flexible options actually work great with dissolvable support material, if you are capable of using it.

Nylon Filaments

Nylon can also be referred to as a polyamide and are generally strong (and often semi-flexible) options for 3D printing. Almost all nylon options offered on the market are more expensive than PLA and ASA, but they are best for many applications. If you really require impact resistance, you are likely going to want to with nylon due to its durability to flexibility ratio.

The strongest material I have ever tested to date is actually a Nylon. It is PolyMide CoPA by Polymaker, and I am not just saying that because they have contributed a chapter to this book. You can view my video “Nylon Comparison Part 1” on YouTube to see exactly how I have come to this conclusion. If you need strength – check this material out. That said, it is difficult to get a clean surface quality with that particular material.

Nylons come in a wide variety of options, but almost all have a bit of difficulty sticking to build plates without extra help. PolyMide CoPA has actually been designed to reduce warping. As mentioned elsewhere in this book, you will likely want to grab some Elmer’s glue, do a 1-1 mixture with water, and lightly brush it onto a clean glass build plate. After evaporated, nylons will stick great and pop off fairly easily with a scraper. You can also use a glue stick if that is your preferred method.

In fact, Magigoo now makes a nylon option. Though I haven’t tried it yet, if it works half as well as their other options, I would think it would be great to have nylon parts not warp.

For the example below I will be using PolyMide CoPA by Polymaker since it is my current favorite nylon material, but every single nylon will require its own settings.

Some overall notes for nylon filaments:

- Nylon is hygroscopic and must be kept dry. This is not an option as leaving out your nylon spool can actually make it absorb too much moisture in a matter of a day or two, sometimes within just a few hours. You should keep all nylon spools vacuum sealed, and in a dry area (45% humidity or lower). I actually print my nylon while in a dry box as to prevent water ever being an issue, especially on long prints. You should invest in a dry box if you plan on printing with nylon frequently.
- Many nylons have a shelf life and will not print as well after a year of being opened, regardless of how well you store it.
- Nylons are very durable and have an excellent strength to flexibility ratio.
- Nylons often do not come in many color options, but most can be dyed

(since it can absorb moisture). I have tried this in the past to mixed results.

- While Polymaker's options of nylon do not warp a lot, the vast majority of nylons I have tried have been very difficult to print without warping, especially the options made by Taulman. Check out Magigoo meant for nylon if this is a continuing issue.
- Just as with flexible filaments, removing parent support material can be very difficult with nylons. Dissolvable supports seem to work well, but those can be difficult to use and require specific printer setups.

PolyMide CoPA Slicer Settings:

Print Temperature (263°C)

After playing around, I personally came to sticking with a printing temperature of 263°C. Most nylons do not need this high of a printing temperature, and you should refer to the manufacturer's recommendation if not using PolyMide CoPA. Nylon can definitely be finicky with your temperature, so you may need to test a little.

Build Plate Temperature (66°C)

PolyMide is one of the few nylon mixtures that does not require PVA on a glass build plate, but I still do it as to help mitigate any bed adhesion issues.

While PolyMide requires a higher print bed temperature, the majority of nylon materials I have worked with in the past prefer a 45°C print bed, and the vast majority require a glass bed with a PVA mixture to print. Check out Magigoo's line for nylon if you plan on printing nylon a lot.

Check with the manufacturers recommended print bed settings to be sure.

Retraction: (Direct Drive: Distance: 3.5mm – Speed: 35mm/s – Minimum Travel 1mm) (Bowden: Distance: 4mm – Speed: 40mm/s – Minimum Travel – 1mm)

Without going too much into detail, these settings seem to work well with me and this filament. There isn't much to go over here, but you will need to tweak accordingly depending on the nylon you end up using, and if you end up using something like the Hemera.

Speed: (Direct Drive: Speed: 45mm/s - Outer Wall: 22mm/s – Top/Bottom: 22mm/s – Initial Layer: 20mm/s – Travel Speed: 150mm/s) (Bowden: Speed: 40mm/s – Outer Wall: 20mm/s – Top/Bottom: 20mm/s – Initial Layer Speed: 20mm/s – Travel Speed: 150mm/s)

These settings worked for me when working with PolyMide CoPA, but may not be great for other Nylons. Since the CoPA is a decent amount stiffer than

many other nylon blends, you will likely need to go slower than this for other manufacturers.

I don't recall working with a specific nylon that required drastically slow print speeds, so proceed with what I have above, unless you read otherwise by the manufacturer.

Support Settings: Overhang Angle: 50° - Density: 15% - Z Distance: 0.2mm – X/Y Distance: 0.8mm (Disable Support Interface)

These settings stay the same as flexible filaments due to the difficulty to remove the support material cleanly. If you cannot remove the support material and it is stuck to the print, increase your Z distance.

If you notice a lot of drooping and that the particular nylon you are working with is unable to bridge gaps, increase the density. If you think you will be able to remove the filament, go ahead and test out support interface – I just personally haven't had great results in being able to easily clean it all up. Supports settings can be near impossible to hone in on particular nylons, and there isn't much you can do other than try dissolvable supports, or redesign your part as to not have such extreme angles.

NOTES ABOUT PolyMide CoPA and Nylon Alloy 910:

I haven't done a ton of work with PolyMide CoPA, but what I have done has impressed me immensely. It is the only material I have ever done my strength test on that I was unable to break whatsoever. Polymaker has made me a believer when it comes to this material – especially since it is not that difficult to print. That said – since it doesn't use an active cooling fan, it can be difficult for the print to come out cleanly.

Alloy 910 is another favorite of mine made by taulman3D and was a go to for me whenever I really need a part to have strength (until discovering PolyMide CoPA). I have used this material twice now for 3D printed planetary gear skateboard wheels, and they held up to a LOT of impact. They wouldn't break no matter how hard I jumped on my skateboard, and they eventually cracked under the pressure of a few professional skateboards (which can be seen in a couple Braille Skateboarding YouTube video). Compared to an ABS set that broke the moment I jumped on my board, it is clear nylons have their place in 3D printing.

Finding the right elasticity, shore hardness, and tensile strength for your part may be difficult, but there seems to be a wide variety of nylon options on the market today. You just need to think about what you need for your particular

application.

Carbon Fiber Reinforced and Filled Filaments

Carbon fiber reinforced materials are filled with continuous fibers or fiber particles that result in parts with improved physical properties and high stiffness. There is a variety of carbon fiber reinforced options out there for 3D printing, but they all require drastically different print settings. Because of this, I will not be going over particular slicer settings. In general, you will need similar settings to the material that Carbon Fiber is reinforcing. This means when working with Carbon Fiber Reinforced PLA, you will want to use similar settings to that of PLA. The same is true with ABS and Carbon Fiber Reinforced ABS, and so on.

Essentially, you will want to use carbon fiber reinforced materials when you require properties of the original material, but with a higher tensile strength. That said, it seems most carbon fiber blends will have less layer adhesion properties. This is kind of unavoidable though many new options seem to try to improve this layer adhesion. Almost all carbon fiber blends print with a great surface quality.

Carbon Fiber Reinforced ABS has the high glass transition temperature of ABS, yet is much more stiff and strong. It is also easier to print while still being able to be acetone vapor smoothed. Carbon Fiber Reinforced Nylon is an awesome combination, one that is perfect for a wide variety of strength and heat resistant applications. You just need to make sure you really zero in the layer adhesion.

Personally, I have never found a use for Carbon Fiber Reinforced PLA, since it turns out more brittle than standard PLA (due to a lower elongation), but I am sure there applications for everything.

You will HAVE to work with hardened steel or ruby tipped nozzles when working with carbon fiber nylons, since the carbon fiber will add abrasion and lead to your brass nozzle being worn out rapidly fast. A hardened steel or ruby tip nozzle will have a higher hardness than the carbon fiber blend and should not wear down your nozzle (or it will at least take much, much longer). I have standardized to using Nozzle X by E3D, though they now have a newer option called ObXidian that I have yet to try.

I am not going to include specific slicer settings for carbon fiber blends because they all vary drastically depending on what the carbon fiber is blended with. Just please note that as of now, generally all carbon fiber blends have lower layer adhesion properties.

Quality Options

The quality, and amount of time to print, will vary based off of a few factors, but two in particular have the most effect – your nozzle diameter and the layer heights you are printing at. Below we will take a look at a few of these options.

Nozzle Diameter



The nozzle diameter will determine the line width of your print segments, which will affect the tolerances in the X/Y direction. While many people prefer to slightly tweak their line width from their nozzle diameter – I normally keep it the same. I have been experimenting with increasing by 10% to good results (as in printing 0.44mm line width with a 0.4mm nozzle). Any part of your print that is thinner than your line width will not be printed, so you can imagine how a thinner nozzle diameter can lead to a higher quality print. This isn't exactly true anymore, since Cura now has the ability to "Print Thin Walls", though they will not be to proper dimensions if thinner than your line width.

The biggest issue with reducing your nozzle diameter and line width comes with the print time required. The fact that you have to slow your print speeds down to prevent bottlenecking, that you have to print at lower layer heights, all along with the actual lines being thinner, your print can take exponentially longer.

The general rule of thumb is to allow for a clearance of $\frac{1}{2}$ the nozzle diameter for parts that mate together, though as you see in the "Parts Not Mating Together" chapter - it is smart to print your own tolerance test to see what your clearances should be. You should be able to print with tighter clearances when using a thinner nozzle diameter, since $\frac{1}{2}$ the nozzle diameter will be a smaller number.

One other issue with printing with a small nozzle diameter is the fact you will likely have less layer adhesion. As covered in the "Poor Layer Adhesion" and "Material Science" chapters, you get increased entanglements between your layers when using a larger diameter nozzle.

When printing with a very small nozzle you will need to use a geared extruder. You need the proper amount of torque to push through 0.15mm or 0.25mm diameter nozzles due to the extreme bottlenecking. It is also smart to do this on a direct extruder versus a Bowden, since most Bowden setups will have a rough time pushing filament through an extremely fine diameter

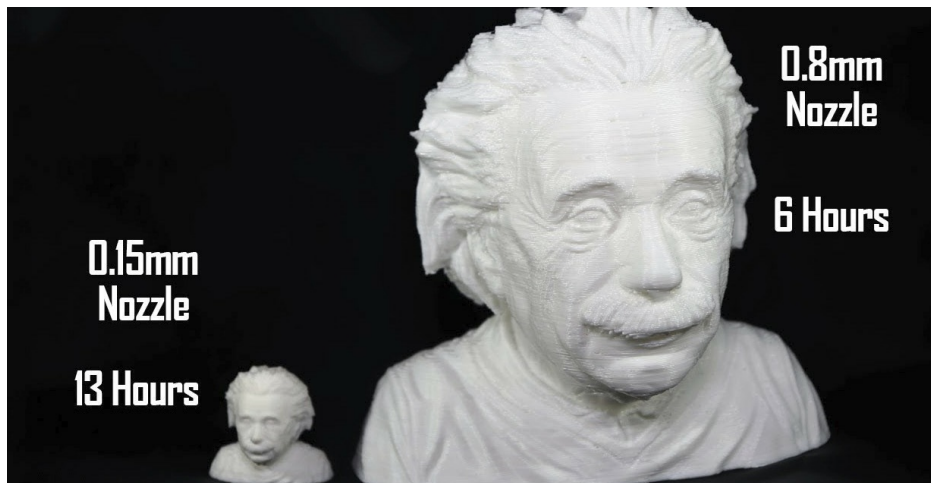
nozzle.

Personally, I have standardized to using 0.25mm, 0.4mm, and 0.6mm nozzles. It seems that the 0.15mm nozzle is very hard to dial in and takes an extraordinarily long amount of time to print with, and the 0.8mm nozzle is just too low of tolerances for what I am looking for. The only time I have used a 0.8mm nozzle is when printing in vase mode. I have printed with a 1.4mm nozzle on an E3D SuperVolcano, but that was only to test for a video, and I have no real applications for it. These would be great for extremely large printers when your quality is not super important.

I have a couple of videos going over printing in different nozzle diameters at my YouTube channel – The 3D Print General – if you would like further information. The most recent one is titled “3D Printing with Extremely Fine Nozzles” and it covers a ton of information on quality options and limitations in FDM 3D printing.

If you require super high detail, such as small jewelry, I would suggest checking out resin printing instead of FDM.

How the nozzle diameter effects layer heights



As stated elsewhere in the book, you have a range of layer heights that will result in reliable prints based off of your nozzle diameter. Essentially, you want your layer heights to stay within 25-75% of your nozzle diameter. This means a 0.15mm nozzle should print roughly within 0.04mm – 0.11mm layer heights, and a 0.8mm nozzle should print within 0.2mm – 0.6mm layer heights. Some printers advertise a 0.05mm layer height capability, though I would not personally suggest going that low when using a stock 0.4mm nozzle.

When you go outside this range, the extrusion reliability and quality will often go down. When you try to print on a small nozzle with too large of layer heights, you will surely clog and grind filament more frequently, and when you try to print too low of layer heights on a large nozzle, you won't be printing at quite the tolerances and quality that you could with a proper nozzle diameter.

Layer Heights

Layer heights refer to how thick each individual layer is in the Z-direction. Large layer heights reduce the quality in the Z-direction, but allow for a much faster print. When printing at the same speeds with same nozzle diameter, a print that is 0.2mm layer heights will take half as long to complete as the same print with 0.1mm layer heights, since it will have $\frac{1}{2}$ the amount of layers.

It seems that the speeds you can print at with a standard extruder/hotend setup works on a bell curve. You need to slow down your print as you go to very low layer heights and when using a small diameter nozzle in order to prevent bottlenecking and nozzle clogs. You also need to slow down your print speeds when going with a very large nozzle with large layer heights in order to get the proper viscosity. If you print too fast with large layer heights and nozzle diameters, the particular material may not have enough time to melt.

For example, the standard E3D V6 hotend is advertised at printing up to 15mm³/s, meaning that is the maximum amount of volume the hotend can reliably extrude per second. You can print a larger volume of material per second with a different type of hotend - such as the E3D Volcano. The E3D volcano advertises printing volume up to around 40mm³/s – meaning you can print much faster with larger nozzles and layer heights, assuming your extruder can handle those speeds.

It seems I can print with the fastest linear speed on my standard V6 setup with a 0.6mm nozzle at around 0.25mm layer heights. Once I bump up to the 0.8mm nozzle I need to slow down my print speeds, and the same is true when moving to a 0.4mm or 0.25mm nozzle. The larger nozzle will likely still allow the print to finish faster even with the lower print speed due to the additional volume of material that is being deposited with each move.

If you would like to read further information on these limitations with hotends, you should refer to the “Speed Limitations” chapter in this book.

For over 90% of my prints I have standardized to a 0.4mm nozzle. I have this nozzle in hardened steel and will print the vast majority of my prints at 0.1mm – 0.25mm layer heights with this 0.4mm nozzle. This will work for the majority of 3D printing applications.

Limitations with 3D Printing

3D printing is not a perfect end-all solution for manufacturing parts, especially with today's technology. The RepRap project started as a dive into the unknown. Problems were addressed as they came and prints were considered a success if they were useable at all. These early prints would be considered unacceptable in today's terms.

Most machines are not plug-and-play and require a bit of knowledge before you can even have one successful print. I'm not sure they will ever be to the point where you need no background knowledge to use.

Almost all 3D printing manufacturers, especially those on Kickstarter, will brag how their printer can be instantly used by anyone without any knowledge for high quality parts, and that they are doing something revolutionary - but there really isn't a desktop machine like this. Every single printer will have its own limitations and difficulties.

Hotend limitations

Most of you should really read this section, since I assume the majority of you bought a Creality machine or a clone of one. These inexpensive machines will almost always come with a hotend that is not all-metal, and they have some PTFE tubing going all the way into the heaterblock.

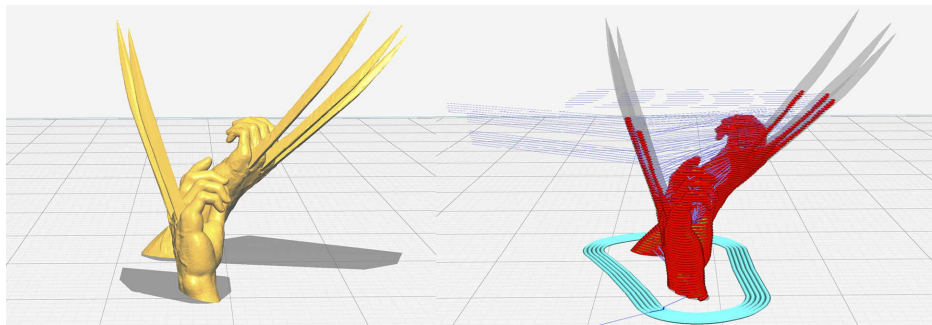
These hotends are going to be limited to reliably print at a max of 240°C, though some may advertise 10 degrees hotter. This means you aren't going to be able to print anything that has a printing temperature above 240 degrees. I actually prefer to print ABS above 240 degrees, so these hotends can really be limiting.

I always suggest using an all-metal hotend, such as ones made by E3D, Slice Engineering, MicroSwiss, or another reputable brand. These hotends can get to 285°C reliably with a standard thermistor, and some can actually get up to 500°C if you swap the thermistor to a thermocouple and MOSFET. 285 degrees opens you up to so many more material options, and 500 covers the gamut.

Unprintable models

I go over this in further detail in the “Model Errors” chapter, but the models you can print are limited to the capabilities of your machine, and 3D printing in general.

Walls and details that are thinner than your nozzle diameter/line width will not be shown on your final print. In the example below, Wolverines claws get thinner than the line width I am using. This means the printer will not recognize them, and your final print will look like the image on the right. If something is thinner than the thinnest diameter nozzle available, you have a part that cannot currently be FDM 3D printed. Well that isn't exactly true anymore - Cura also now has the ability to “Print Thin Walls” in the “Walls” section. This may actually make it so your printer will recognize and print these walls. That said – there is no real way that they would be to the proper dimensions, though it is definitely an improvement over older slicers.



If you need text to show up on a side of a print in which the thickness of each letter is less than 0.1mm, you may not be able to find a desktop 3D printer on the market that can achieve this detail. These small models and details are why SLA printing is such a useful tool. I highly suggest getting into SLA resin printing if you plan on needing extreme detail and very small parts. This edition of this book now has a chapter covering the basics of resin printing.

There are also issues involved with the need of support material. Since gravity will always be a factor, FDM printing will have problems printing clean overhangs. Not only will cleanliness of the underside of overhangs be effected, but you will not be able to print specific models since you cannot remove the support material.

You essentially will not be able to print any object entirely hollow unless it is a sphere or comes to a rounded top. If you want to print a giant rectangular cube, it will require at least 10% infill in order to have the top surface print cleanly. A true hollow, large rectangle would require support material, but there wouldn't be any way to remove it, meaning you cannot practically print one.

Active patents that prevent innovation

One of the things that prevented 3D printing from reaching the masses sooner were a few integral patents. As with many blossoming industries, unused or all-encompassing patents that have a long life span can prevent people from innovating out of fear of being sued. Now that 3D printing has grown into such a lucrative market, there are countless patents that currently exist, though luckily they have been expiring over the last couple of years.

Some of these patents prevent innovation to enclosed build environments, some to build plates, and others that encompass entire sections or methods of printing. Many of these patents are actually expiring, which helps to contribute to the amount of inexpensive machines that are popping up on the market. Days where you were forced to spend \$2,000 on a basic machine are no more. I expect there to be fully enclosed, actively heated printers to hit the consumer market at around \$1,000 any day now. This would open up higher temp material options to the masses.

There may be innovations to these patents that still exist that are not currently even explored. We will hopefully see further innovation as these patents expire over the next 5-10 years.

Print speed limitations

Barring a major change in the process used in FDM printing, there will always be a limitation to just how fast a part can be printed. The better the tolerances, the longer the print - there is no way around that. That said – the new Voron builds can get to speeds I didn't think reasonable, but even these have limitations.

If you require speed when 3D printing, you may want to consider an alternative 3D printing technology such as MJF or DLP over FDM. While significantly more expensive, these technologies were developed to drastically speed up printing and alleviate some other process limitations found in FDM.

Please refer to the next chapter titled “Speed Limitations” for an in-depth explanation.

Speed Limitations

Printer manufacturers will often advertise print speeds that are either not really possible, or will result in a subpar quality print. There are limitations when it comes to printing fast, many of which have physical impossibilities.

There is no doubt that printing at larger diameter nozzles with larger layer heights will result in a print that completes in a shorter time than their counterparts, so this chapter will not really be covering this. This chapter is in relation to the speed your extruder carriage is moving and the amount of material it is attempting to extrude.

I do have to comment though that printers lately have actually been impressing me with just how fast they have been getting. Thomas Sanladerer finished building an amazing Voron 2.4 printer on his YouTube channel which he was able to print at a top speed of 1,000mm/s and accelerations of 20,000mm/s/s. While he prefers to print at half that speed, that is still absolutely unheard of for most modern printers, as you will hear me explain in this chapter. These Voron builds are pushing the limits of 3D printing and are really exciting, but I am covering standard desktop printers in this book you are likely to buy. If you are interested in building one of these awesome printers, you can see his video by searching the title, “My new favorite 3D Printer! The Voron 2.4 Build Experience” on YouTube.

****Note**** Please keep in mind I always suggest erring on the side of printing slower, as you see throughout the rest of this book. I am just going to be going over the limitations involved with printing top speeds in this chapter.

Nozzle diameter limitations

This is a little vaguer than the rest of this chapter, but the diameter of your nozzle will limit you on just how fast you can extrude. This is due to bottlenecking between the extruder and the nozzle.

Just as with traffic when driving, attempting to squeeze material through a very tiny hole will have its own limitations in speed. It is difficult for me to give exact top speeds on this, but the smaller the nozzle diameter, the slower you are going to have to print to prevent bottlenecking. While I am able to print just fine with a standard V6 hotend on a 0.4mm nozzle at speeds of up to 100mm/s, I have increased difficulties going with a smaller diameter than this.

Pushing 1.75mm filament out of a 0.15mm nozzle diameter is going to have a lot of this bottlenecking. With this extremely tiny nozzle diameter, I am forced to go down to just 20mm/s print speed. Any faster than this will have a lot of difficulty overcoming bottlenecking, resulting in a very under extruded print, if it doesn't just result in extruder skips or a nozzle clog. Extruders such as the Hemera will help with this, but there will always be some form of limitation due to this bottlenecking.

A larger diameter nozzle will allow for faster extrusion without this bottlenecking occurring, though you will need to read the next two section to understand how that has its own set of physical limitations.

Hotend limitations

This is a physical limitation that is impossible to avoid. A hotend can only extrude so much volumetric material per second. The material needs time to heat up and become viscous enough to actually come out of the nozzle.

Many hotends have a rating for just what volumetric throughput it can handle. The standard V6 hotend by E3D – likely the most common hotend on the market - has a max throughput of $15\text{mm}^3/\text{s}$. This maximum throughput will change depending on the material you are using, but the maximum it can handle with perfect conditions is roughly that $15\text{mm}^3/\text{s}$.

Based off of that max rating, you can do some math to figure out the max speed the hotend can handle depending on the line width and layer height. You can essentially figure out how much volume of material is coming out per mm travelled. There used to be a great calculator for this online by Print Industries, but it seems the website is no longer around. I am currently unable to find a good replacement, but you may be able to find a calculator for your maximum speed for your particular hotend, layer height, and line width.

Based off of this $15\text{mm}^3/\text{s}$, the max speed for 0.2mm layer heights on a 0.4mm layer lines would be 240mm/s. This is likely far higher than you would ever want to go for your 3D prints, but you can at least see the physical limitations involved with a hotend. When bumping these numbers up to 0.8mm layer lines at 0.4mm layer heights, this maximum speed is down to 60mm/s (due to more volume being pushed out the nozzle per mm travelled).

Mind you those numbers are pushing the hotend to the max. I would error on the side of going for roughly 50% the maximum throughput rating for your hotend, especially when not printing in PLA. This means that with a 0.8mm nozzle at 0.4mm layer heights on a standard V6 hotend, you likely shouldn't go above 30mm/s speed. This is when your hotend will really limit your maximum speeds.

If you plan on only dealing with a 0.4mm nozzle, then your V6 hotend will not really be limiting you on your maximum speeds. It is when you want to work with these larger nozzles that it will become a factor. And that is why hotends like the E3D Volcano and SuperVolcano exist.

The Volcano allows for up to 3x the maximum throughput of the V6, and the SuperVolcano allows for up to 11x the maximum throughput of the V6. These are the hotends you want to use if you want to print big parts with large layer heights. This way your hotend will not be the limiting factor on you printing fast.

So, if you ever see a printer manufacturer advertising print speeds of 150mm/s+, you need to question the capabilities of the hotend, and with what nozzle diameter and layer heights they are referring to.

Pictured below is the E3D SuperVolcano hotend. You can see just how large the melt zone is.



Please refer to my video titled “Taking it to the Extreme with the SUPER Volcano” for more information on hotend limitations and testing out these higher rated hotends.

Extruder Limitations

As covered in the “Extruder Stepper Skipping” chapter, your extruder itself is going to be limited to how fast it can push out filament. If you are using a non-g geared extruder, especially one on a Bowden setup, you will never come close to printing at speeds where you actually require a higher rated hotend. When I am using an inexpensive printer with a Bowden non-g geared extruder, I am really limited to just about 40mm/s max speeds when printing with a 0.4mm nozzle at 0.2mm layer heights. Any faster and I will hear that annoying clicking of the extruder motor skipping. This isn’t true across all Bowden setups, since many of you may be printing 60mm/s+ without any issues. I personally like to stick to that 40mm/s for stock setups just to be safe.

When going with larger layer heights and larger nozzle diameters I am going to be forced to go much slower, due to the extruder motor needing to push out more volume per mm travelled.

Ever since switching to the Bondtech BMG on a direct extruder setup, or the even better Hemera style extruder and hotends, I have never faced these limitations. I am sure there is a point where the extruder needs to spin too fast, but even when using the SuperVolcano with a 1.4mm nozzle and 1mm layer heights, I was still able to print 55mm/s without any issues, but my extruder gear was spinning extremely fast. You need to know the limitations of your extruder setup.

Acceleration/Deceleration

If you hear that someone is printing at 300mm/s, you can almost be guaranteed they are never actually reaching that top speed (with the obvious exception of the Voron build that Thomas Sanladerer did mentioned at the beginning of this chapter). This is because the hotend needs time to accelerate to this speed, and then time to decelerate as the print follows its toolpath.

This is the main reason just doubling your print speed is not going to cut your print time in half, particularly on small objects. Your print time is going to have diminishing results the faster you set your print speed, in a pretty logarithmic fashion. This is where your acceleration and jerk settings are going to come into play.

Your jerk settings are the initial speed your hotend will instantaneously start travelling at from a stop. This would mean if you set your jerk to 20mm/s, your extruder will instantaneously start at 20mm/s, and then accelerate from there.

So if you have a jerk of 20mm/s and an acceleration of 1000mm/s², it will take your hotend 0.1 second to reach a speed of 120mm/s. This may not sound like a lot of time, but you have to factor in that most 3D models require a lot of starting, stopping and changing of directions. And your printer can't just come to a complete stop after travelling 120mm/s, it needs to decelerate, taking roughly another 0.1 second to stop.

When we bump that print speed number up to 300mm/s, we are talking about a time of over 0.25 seconds to reach that speed with the same jerk and acceleration settings. You can imagine how your printer may never actually have a quarter of a second to accelerate and decelerate, meaning you would never actually reach 300mm/s.

You could of course bump up your acceleration and jerk settings, but that is when you are going to come into problems covered in the next two sections.

Frame of your machine

The frame of your machine is going to really limit you on how high your acceleration and jerk settings can be. While Cartesian printers are the most common machines out on the market, they are also going to be the machines that limit you the most on acceleration and jerk.

And this is because on Cartesian printers your print bed is going back and forth in the Y direction. This heavy, large print bed constantly accelerating and decelerating is going to cause your machine to rattle. While this may not be much with low jerk and acceleration settings, the higher you go, the more extreme this will get. When I set my acceleration and jerk settings very high, my entire table was rattling all over the place, to the point things were falling over. You can see just what I mean on my video titled “How Fast Can you 3D Print?”.

You can harness your printer to the print table, but your ghosting problems are going to increase due to the lack of vibration dampening.

This is a major reason people prefer CoreXY and Delta machines. Delta machines are far less common, but the one thing they have going for them is the increased acceleration possibilities. Your printer really shouldn't rattle much at all even with very high acceleration and jerk settings with a Delta machine. The FLSUN SuperRacer I reviewed on my YouTube channel could have accelerations over 3,000 mm/s/s without any issues whatsoever.

CoreXY machines have pretty much become the preferred frame for most makers out there, including myself. The fact the build plate is only moving in the Z-direction means you will experience exponentially less rattling. This lower amount of rattling will not only help to print tall skinny parts and not have them be knocked over mid-print, but it will also help you to achieve higher acceleration and jerk settings without losing quality. The only issue will be when using a very heavy carriage, which is why certain direct drive extruders with a heavy stepper motor may be your limiting factor on acceleration and jerk. I love the Hemera extruder by E3D but it comes with a heavy stepper motor, meaning there will be a decent amount of weight on the carriage.

While I keep my acceleration to around 500mm/s² on my Cartesian machine, I can bump that to over 1000mm/s² on CoreXY without any loss of quality. You can go even higher if you are comfortable with your setup, as many makers do. Remember that Voron build I mentioned by Thomas Sanalader has an amazing acceleration of 20,000mm/s², so it is definitely possible to go higher if you have a well-built machine.

Print Quality

Just about everything covered above are actual physical limitations involved with hitting high print speeds, but the one thing we haven't covered is print quality.

Assuming you are within the limitations of your hotend and extruder setup, a high print speed will not really decrease your print quality that much. The thing that will decrease your print quality the most is high acceleration and jerk settings.

The biggest issue is ghosting, which you can see covered in the “Ghosting” chapter in this book. Ghosting's main culprit is due to high acceleration settings on a frame that is rattling a lot without vibration dampening.

If you don't have a well-built frame, especially one that is Cartesian, you have an increased chance of experiencing Z-wobble, layer shifts, and parts being knocked over. There are likely to be extra artifacts on your print that are hard to explain or fix due to the difficulty honing in your retraction and other settings.

This is why I always suggest going slower in all settings if you are having difficulty getting a clean print. I am sure you would be willing to wait a couple extra hours to be assured your print will come out with the quality you expect. Many makers print much, much faster than I do - but I will always prefer a slow clean print to a fast one with potential artifacts.

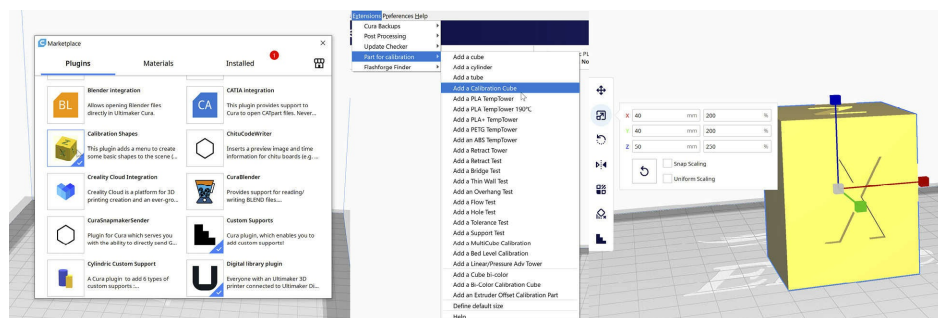
Cura Tricks

I figured I would add this additional chapter because there are a few Cura tricks that are outside the normal settings. Cura, and I am sure other slicers as well, can actually do more than you would think. A lot of what I am covering here I also explore in a few videos on my YouTube channel.

Plugins

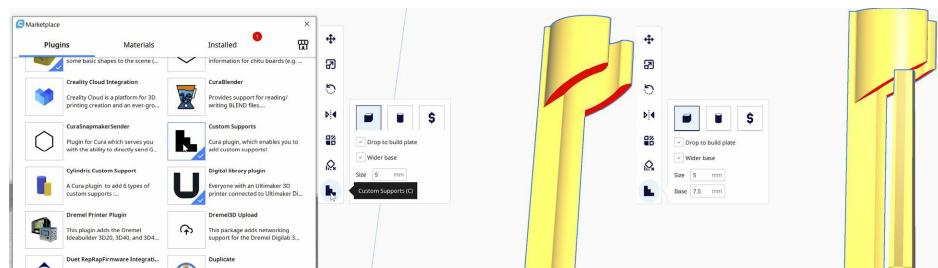
Cura has the ability to add third party plugins, each with its own benefit. To access them, you would click on the top right where it says “Marketplace”. I do not use many of these, but some of the ones I do use include:

Calibration shapes:



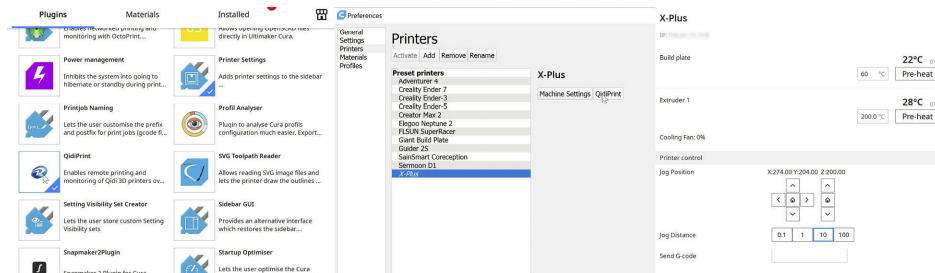
This is a very useful addition since I can now bring in simple shapes. I do not normally use this to print a calibration shape, but I mostly use it in conjunction with combining multiple parts, which I will cover in the next section in this chapter. You can take these shapes and then manipulate the dimensions to fit whatever need you may have. This saves a lot of time from having to create your own shapes or finding calibration tests on Thingiverse.

Custom supports:



Custom supports is very useful to add any support structures to your print that the slicer is not adding. I mostly use this is if I am printing with supports that are only touching the build plate, or have them off entirely, but there is just one overhang that requires it. It can be annoying when Cura adds supports to every hole in your print, even when it isn't needed. So I will often turn supports off, then choose the exact section that should have supports. Sometimes Cura will just miss an overhang for some reason, so this helps in that case as well.

Specific manufacturer plugins:



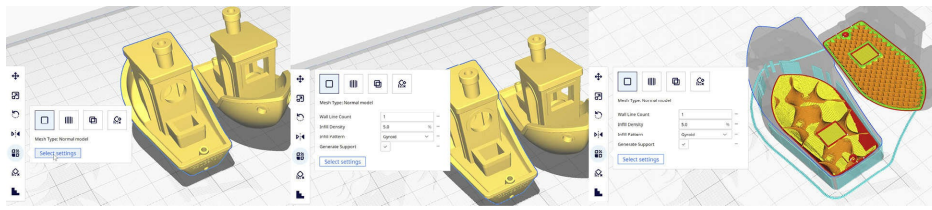
I have the QidiPrint plugin added because I have the Qidi 3D X-Plus printer. This plugin allows me to control the printer while using Cura, since the printer has the ability to connect to the internet. This way I can continue using Cura and take full advantage of controlling the printer remotely. This does require you to not only add the plugin, but also find the plugin online and then add the printers to your Cura folder. I am sure the same is required for your specific machine, but having these plugins allow for using Cura to fully utilize your specific machine.

There are many other plugins that may work for your needs, so just search online for help if you are having difficulties with any of them.

Per Model Settings and Combining Parts for Multiple Processes

This is a really cool feature that many people do not know exists. You can actually bring in two different objects into Cura and then choose specific settings for each, or overlap them and have one model affect the other.

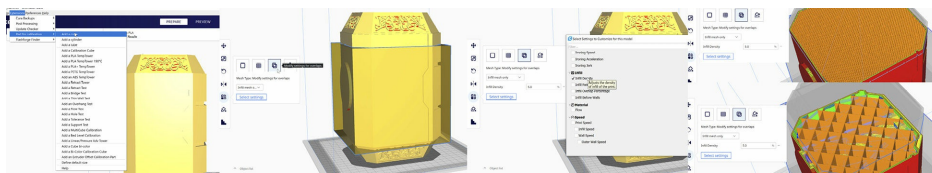
Different settings for models on the same build plate:



This is the most basic use of this “Per Model Settings” that you see on the left side of Cura. Let’s imagine I have two models I need to print and one requires a completely different infill or shell wall count than the other. For this example I am just using two Benchy prints, but the same goes for when you need to print a lot of models and each one requires different settings. Rather than having to print each on their own bed, you can change the settings for just one model.

In the example above, the Benchy on the left has less walls, lower infill density and a different infill pattern than the one on the right. It also has support structures unlike the one on the right. This allows you to save time using one print bed to get multiple models of different settings printed. The one thing you can’t do is change the layer height on one model, but most other settings are available.

Change infill or any other setting in one section on one object by overlapping them:

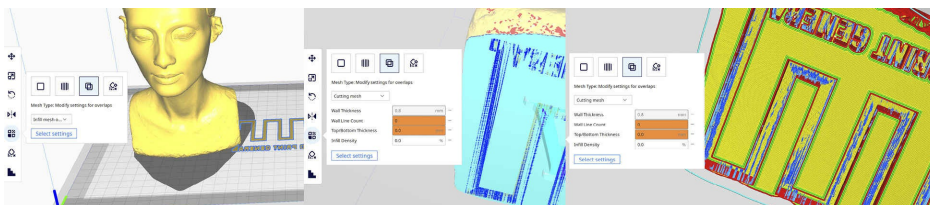


This is one of the times I use the “Calibration Shapes” plugin since it makes it easier to bring in random objects. In the above example, I am printing Mjolnir, and as you can tell, it is solid. This means that it will use a lot of infill material if you were to keep it all the same. Instead I bring in a second

object, a cube in this example, and I scale it and raise it to the middle section of the hammer. In order to raise an object off of the build plate, you will have to uncheck “Automatically drop models to the buildplate” under Preferences >> Configure Cura.

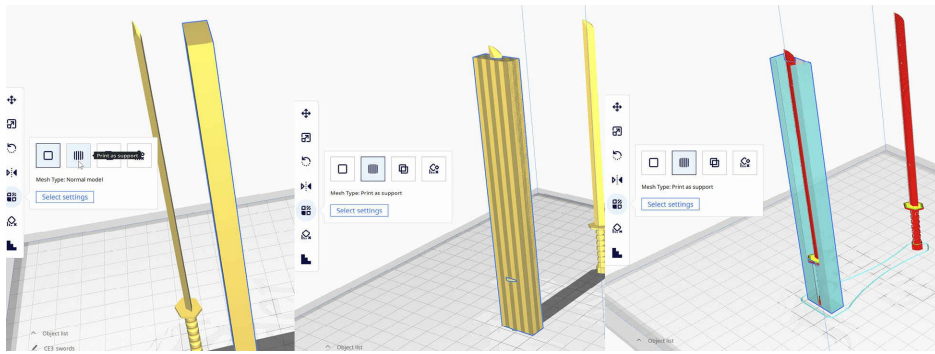
You would then click on the cube and select the “Per Model Settings” on the left side of Cura. Once that pops up, you would then choose “Modify Settings for Overlaps”. This makes your box, or whatever object you bring in, change the settings on your main model. You can select from a list of many settings that you can now tweak on the hammer. For this example, I choose “Infill Density”, and this will make it so only the section on the hammer that is overlapped with the cube has a different infill density. This will enable you to have a high infill density on the top and bottom of the hammer, and a much lower infill density in the middle of the hammer in order to reduce material and print time.

There are so many settings you can change in regard to overlaps that it’s difficult for me to explain them all. You can change the line width on just one section of your print, the extrusion speed for one section, the flow, the support settings, and just about anything else you can think of. Remember though, the object that is overlapping doesn’t need to be a basic shape, it can be anything you want. This means you can add your logo or whatever you want to emboss onto a section of your print.



For this example I am cutting out my Enron logo onto the bottom of this bust. I scale the logo to the correct size and move it to the bottom of the bust where I would like it embossed. You can make it as thick as you would like. I then click on the logo and choose the same sections as the previous example: Per Model Settings and then click “Modify settings for overlaps”. This time though I chose “Cutting Mesh”, instead of “Infill Mesh Only” from the drop down. I then chose all of the settings that I need to emboss, which would be the wall thickness, the top/bottom layers, and the infill. I would then set all of these to “0”. The wall thickness doesn’t matter quite as much in this particular example, but when you set all of those to “0”, the object will essentially not exist, and allow you to cut it out of your main model. This now allows for my logo to be embossed onto the bottom of my print.

Print as supports:



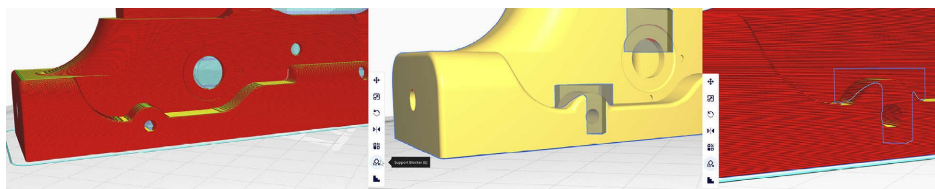
This is similar to the “Parts Being Knocked Over” chapter, but it essentially allows you to set one model to be only supports. I mainly use this when parts are being knocked over and need extra anchoring, but there have been times I have used this in specific case scenarios. For tall, thin parts that may be knocked over or have a lot of Z-wobble due to the build plate rattling back and forth, you can anchor them by bringing in a cube and resizing it to your needs.

You would then choose “Print as Supports” in the “Per Model Settings” on the left side of Cura, and move the object to overlap with your main model. This will now have support structures on either side of the model to help prevent it from swaying back and forth.

Another time I used support prints was when I was printing a water bottle like object that was very wide and had a spout coming out of the top. This model required support structures, but any standard supports would not be able to be removed through the small spout hole. So I brought in some thin cylinders as support, and placed them inside the bottle where it helped the top layers bridge properly. The cylinders could then be broken off and extracted through the spout hole at the top.

These are just some examples of utilizing the “Per Model Settings” and there are countless other ways to play around to get the prints you require.

Support blocker:



Support blocker is another option on the left side of Cura and it allows you to place cubes on specific areas of your print that you don't want any support. It seems that Cura likes to add support structures to holes in your print, even though PLA can easily print a hole without the need for any. So for the example above I am printing in Polymax PLA and since adding support

structures to each hole not only adds to print time, it can be difficult to remove some of these supports as well.

By placing a block in each hole after clicking on “Support Blocker”, it will remove that support material from where you don’t want it. I use this feature mainly for removing support structures from holes on PLA prints, but it can be used for when Cura is placing supports in areas that aren’t needed.

Summary

It seems that Cura has quite a lot of features that aren't explained or explored thoroughly, and I only covered a few in this chapter. Hopefully what I did explain sparked some interest to try out unique settings that will allow you to print in ways you didn't think possible for your project.

I have yet to discover all that Cura has to offer and these tips are things I have only discovered over the last year or two.

Important Accessories and Replacements

You should always have a few accessories and replacements on hand so that you do not need to order parts every time something breaks down. Some of these tools I use every single day, while others are needed a lot less frequently.

You can visit 3DPrintGeneral.com for more parts that I suggest and have personally used, with direct links on how to buy them. The majority of these parts are the same as previous editions of this book.

Accessories

I consider all tools that are frequently used on 3D printed parts and the machine itself as a 3D printing accessory. I actually have a video similar to this chapter titled “Mandatory Accessories for a 3D Printer” on my YouTube channel if you wish to see more, though that video might be a little dated.

Pliers: I can’t think of anything more important to remove support than a set of pliers. You will use these just about every time you print with support. I also suggest getting a pair of slip joint pliers. They won’t be used to remove support material, but they are great for holding onto things. I use a pair whenever I swap nozzles to hold the heaterblock in place while I unscrew the nozzle.

Razor Blade: Cleaning up a print will always be easier with some razor blades. I use them just about every time I print with a brim, and it also helps to clean off any “hairy” residue and pop off support interface.

Model Cutters: These sharp, thin scissors are a lifesaver when it comes to printing very thin parts. If you have a very thin part of your print that is being held up via support material, just going at it with a set of pliers will likely break that fragile area. Using model cutters will allow you to cut this support material off without damaging your print.

Scraper: A scraper will come with just about any 3D printer you buy, but it is smart to have one that you prefer. I like a very strong metal that tapers to a thin, flat front. I have tried out flimsy scrapers that will bend on difficult parts, as well as thick scrapers that are difficult to get under your print without damaging it. Get a strong metal scraper that tapers at the very front. I suggest getting the set by AMX3D.

Metric Allen Screwdriver Set: Just about every printer on the market uses a lot of Allen screws. Having an Allen wrench set is smart, but having an Allen screwdriver set is even better. It can be difficult to get to some of your screws via a normal Allen wrench, but is far easier with an Allen screwdriver.

The most common size of screw used is an M3, but it is smart to have an M2.5, M3, M4, and M5 Allen screwdrivers. I don’t know of any 3D printers that use imperial sizes, almost all use metric. In order to make any fixes or repairs to your printer, you will need this screwdrivers/wrench set.

If you have a very thin Allen key (M1.5), it is actually very useful for pushing out blocks or clogs in your hotend as well.

Solder Set or Solder Seal Wire Connectors (along with wire strippers): Wires become frayed, and you will often need to cut them to attach new

parts. There are plenty of times in this book where I go over how to fix parts on your machine, and most of them require to you cut and fix frayed wires.

The most common way to do this is to get wire strippers, solder, a soldering iron, heatshrink, a heat gun, and maybe some tin. There are plenty of tutorials on how to solder wires, which involve stripping the wire, adding some tin to help solder stick, then using your soldering iron to melt the solder to the two exposed wires. This makes a strong connection which you then cover with some heatshrink so that no wires are exposed. I personally use a wireless soldering iron by Iso-tip since I prefer it that way, but just about any soldering iron would work.

I discovered solder seal wire connectors a couple years ago, which have become my new favorite way to do this. With solder seal wire connectors you do not need any soldering iron, solder, tin, or heatshrink; just wire strippers. These connectors have solder in the middle where you push the two stripped wires into. You then use a heatgun (or even a lighter if you are careful) to melt the solder and shrink the wrap around the wires. I definitely suggest having a set of these for when you need to connect wires if you do not want to pay for a full soldering kit, or are just not good at soldering like myself.

You will still need a soldering iron and solder to connect wires directly to parts, such as with a heated build plate, but these connectors work great anytime I need to connect two wires.

Tweezers: I have a very thin pair of tweezers that come to a point that I use to grab material off the nozzle during the print. Sometimes a print will start and still have some oozed material stuck that is missed, and it is convenient to be able to grab it off really quickly with a thin pair of tweezers.

Zip Ties: Obviously zip ties are not ideal for important parts on your machine, but they can definitely help out in a pinch. They can also help to clean up your wires by tying them all together or to hold something out of the way while you work on the printer. I always have a large box of different sized zip ties on hand since I use them quite frequently, and never know when I will actually need them.

Calipers: Calipers help immensely to accurately measure parts and filament. If you are doing any sort of designing, you are definitely going to want calipers. They also help to properly figure out your E-steps.

Multimeter: A multimeter has a ton of features and uses, but I mostly use the continuity setting for the majority of my 3D printing needs. You can check voltage, amperage, and much more, but just figuring out if a wire is working properly is its best function.



You can easily touch a metal tip to one side of your wire, and the other metal tip to the other side, and you will be told if there is continuity with a beep. This can help immensely when trying to diagnose what isn't working on your printer, as mentioned in many chapters.

You can also check the VREF of your stepper drivers with a multimeter, as covered in a few chapters in this book.

Loctite Super Glue Gel: This can be very handy to have for whenever you need to do minor repairs to any broken 3D prints. Keep in mind that combining different types of materials may require different adhesives, but Loctite Super Glue Gel works wonders on PLA, and it dries extremely fast. You will have to remove the outer container when running low, since there is still about a quarter of the gel left in the tube after you are unable to get any more out.

White Lithium Grease: There are different preferences when it comes to lubing up important rods on your machine, but my favorite is white lithium grease. Periodically lubing both threaded and smooth rods on your printer will help to get consistent, clean prints.

Wire and Nylon Brushes: Nylon brushes are key to cleaning up a dirty nozzle and heater block. The only issue is that the nylon may melt on hot heaters. I really like to use brass brushes, but the added abrasion is not great for the hotend. If you have a very dirty heater block and nozzle, use a brass brush with a heated hotend, but try to use nylon brushes when possible.

Heatsinks: As mentioned in the “Stepper Motors Overheating or Malfunctioning” chapter, it is very smart to have some heatsinks on any parts that may be getting too hot. They help to disperse heat and are very smart to add to the extruder, X, and Y stepper motors. Use with a thermal transfer adhesive backing or paste to attach. It is not always needed but it is an easy method to keep parts cooled slightly.

Something for organization: This is pretty vague, but keeping your

important accessories organized in a place near your machines will help a lot. I personally have a strong magnetic bar by my machines that have all of my tools attached to it. This way when I need something fast I can easily grab an Allen key, pliers, tweezers, scraper, etc. The Ender 3 V2 has a nice little drawer in the bottom of the printer which is very convenient for keeping important tools in that you may need to access quickly.

You can also use some small containers that are labeled, just make sure they are all right next to your machine and organized so you don't go crazy looking for that part you need right away.

Fire Extinguishing Ball: This is something I can't stress enough to have in case of emergency. Hopefully it is never needed, but if it is, you will be happy you have one.

I have read too many stories about someone's printer catching fire and sometimes bringing down the entire house. If you are using the proper electronics on a well-built machine, a fire should never happen, but you want to idiot-proof yourself for something this serious. I have an AFO Fire Extinguishing ball mounted above my printers, and considering it will act as insurance on your house, \$40 is well worth it. Make sure to check out the "Electrical Safety" chapter for more information.

Replacements

While it is nice to have replacements for every part on your machine, this is clearly not practical. Below are a list of inexpensive parts that should be held on reserve in case you were to ever need them.

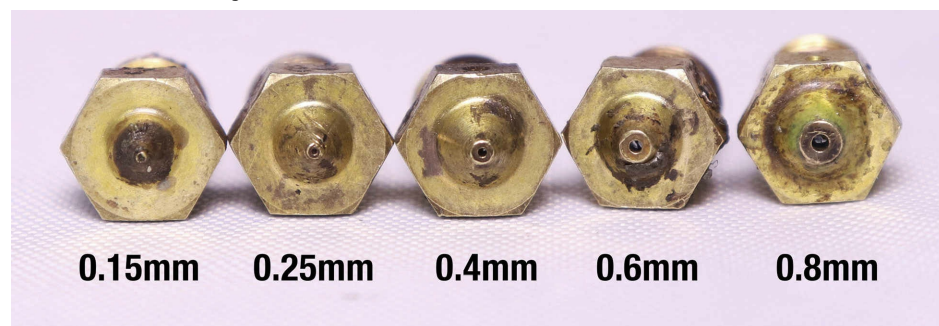
Thermistors: Thermistors essentially act as thermometers for your hotend, making sure the heater sets to the proper temperature. These are very fragile parts, and many heater blocks just hold them in via a small screw and washer. These are very inexpensive and nice to have in case you ever get an “ERR: MINTEMP” or negative temperature readout.

I have gone through dozens of thermistors over the years, and they only cost around \$5, so I definitely recommend a few spares for your hotend.

Heater: Heaters do not need to be replaced very often, but it is another inexpensive part that may come in handy. It can help to save you time diagnosing any hotend heating issues.

Nozzles: This is probably the most important replacement to have on hand. Brass nozzles wear out quite easily, and even the hardened steel ones will eventually need to be replaced. If you are ever having difficulty diagnosing why your prints are coming out ugly, it is well worth swapping out your nozzle. Brass nozzles cost under \$5 and hardened steel ones are closer to \$15, but you will be extremely happy once you change your nozzle and it fixes your problems. I believe hardened steel versions are worth it, since you will not need to swap nozzles nearly as often. Personally, I have standardized to Nozzle X by E3D, though they have a new nozzle called ObXidian that I haven’t used yet as of editing this edition. Be sure to only buy from reputable manufacturers if possible. I buy my E3D nozzles directly from E3D, since tolerances on these nozzles is very important.

I have a video titled “The Importance of Replacing Nozzles” which goes over this further if you wish to learn more.



It would also be smart to have a few different nozzle diameters on hand. A good set would include a couple 0.25mm, 0.4mm, and 0.6mm nozzles, and

maybe a 0.8mm for large prints.

Fans: Brushless fans can become damaged if accidentally hit when spinning, and may just burn out over time. If your barrel cooling fan is damaged, you are destined for repeated clogs. Make sure you have a couple of spare fans on hand for when you need them. The majority of fans on a printer are 40mm x 40mm x 10mm, but some use different sizes. E3D setups use a 30mm x 30mm x 10mm fan on their barrel. Just make sure you purchase the correct voltage for your machine. Most printers made today are 24V so make sure to double check before buying a new set.

Silicone Socks: These silicone socks go over your heater block and will help to prevent black spots on your print and to maintain print temperature. I definitely suggest to always print with these silicone socks, and to have a couple backups. These will eventually degrade and break over hundreds of hours of printing.

Teflon (PTFE) Tubing: PTFE tubing is used to help guide your filament to the hotend. It helps to prevent filament tangling and to make sure everything is guided correctly without any filament snaps. Every so often these can be damaged when pushed into your hotend, especially when using a hotend that isn't all-metal, so having some extra tubing on hand is a smart idea.

Endstops: Endstops are another inexpensive part that will every so often need replacement. This doesn't necessarily have to be purchased before you need it, but it can save you a couple of days for shipping when you run into an issue.

New Innovations

Since it has been two years since my last edition, I figured it would be smart to cover some of the advancements and where the industry seems to be heading. What is exciting about an industry such as 3D printing is that it is constantly changing, and changing fast. Just about all of my main workhorse printers are different now than they were in the beginning of 2020.

Now I do not claim to be up to date on every single innovation, but here are some things that have excited me.

Extruder upgrades

If you have read my 2020 edition, you may have seen that I was excited about the potential of the new Hemera (formally “Hermes”) extruder and hotend combo by E3D (pictured below). Well not only have they become my main workhorse extruder, there are now a couple of other options as well.



As I mentioned in the introduction, the three things that really make 3D printing unique are the material options, the slicer, and the extruder/hotend. Most other parts and functions have been used in other industries before 3D printing became popular.

This means that the extruder is extremely important. I now highly suggest to anyone who wants to print more than just PLA as a random hobby that they use an extruder with a gear ratio and a dual drive feature.

Having a gear ratio means there is less torque required on your extruder due to a mechanical advantage, which I cover in the “Extruder Motor Clicking” chapter. This is not only important for reducing clogs and improving print quality, but is crucial if you want to print more difficult materials such as flexibles and nylons.

Dual drive means that the filament is gripped and moved by two hobbled gears pinching it, rather than just one. The standard cheap extruder that comes on most Creality machines and Ender 3 clones not only has no gear ratio, it pinches the filament between a hobbled gear and a smooth bearing. This means the filament is only gripped on one side and will not be nearly as precise as one that has both sides driving the material. You will have more filament slips and inaccurate extrusion. It may not be noticeable on specific small prints, but it will definitely be preferred in the long run.

Bondtech was the first dual drive extruder with a gear ratio that I had used, and I loved it as you can tell from the previous editions of my book. The

Hemera takes that concept and makes the distance from extrusion to the heaterblock nearly nonexistent. As I have explained elsewhere in the book, extruders can be set up in either a Bowden or direct fashion. Direct extruders are normally preferred because limiting the distance between the extruder and hotend means you can print far more flexible options and at a much faster speed. By E3D eliminating this gap to basically a few microns means there is almost no worry in printing soft, flexible materials.

I have now printed Ninjabflex, one of the hardest soft materials to print, at a whopping 100mm/s without any issues to the print quality. This was unheard of before the Hemera, as I was maxing out at around 25mm/s previously. I personally would never print 100mm/s to ensure quality, but the proof of concept is clearly shown with this extruder/hotend combo.

Since the release of the Hemera there are now a few other options including the BIQU H2 that I reviewed on my YouTube channel, which takes the Hemera idea and then drastically reduces the weight.

Bondtech also seems to have their own version, though I have no experience with it. There is also the Drop Effect Omnia Drop which I also like. Just keep in mind that there have been massive upgrades over the crummy \$5 extruder that your Ender 3 came with.

Please note that I do cover one issue that may be currently caused by dual drive extrusion – which is covered in the “Patterns in Outer Surface” Chapter.

Slicer options

This is an area that I am not going to cover quite as much, due to my inexperience with the other current popular slicers. You will see me reference Cura throughout this book, because I had always preferred it over the other popular options available years ago. Currently there are quite a lot of new options that many makers prefer to use.

If it seems like Cura isn't getting what you need done, or it isn't quite as user intuitive to you as it is to me, then I can now highly suggest checking some of these out. The most popular free competitor is likely PrusaSlicer. Just like with Cura being designed for Ultimaker, PrusaSlicer was meant for Prusa printers but you can slicer for just about any printer that you need. With Josef Prusa always being on top of his game, the reviews are great.

Another one that I keep getting comments on that individuals seem to like is Idea Maker by Raise3D. As with the other two options, this slicer can be used for just about any printer and many people like how user intuitive it is. Becoming so entrenched in this industry by printing dozens of parts a day for years with only Cura, I may not recognize what is most user intuitive to someone new to 3D printing. If Cura seems overwhelming, then I would at least suggest watching a video or two on these two options as a free alternative.

Price reductions

One of the best innovations in 3D printing has been the reduction in price. While not having anything particularly unique to it, the Ender 3 has become the most popular printer for a reason – it works and it is cheap. \$200 for a printer that actually works? That was absolutely unheard of when I got into 3D printing.

If you saw the second video I ever made for my YouTube channel, I built a “reliable” printer for \$300. It took me over 8 hours to build, it was smaller than the Ender 3, and just about every spec on it was worse than that Creality printer. So we have definitely come a long way.

While Ender 3’s are definitely great for their price, you are likely still going to want to do some improvements if you want to turn the printer into a workhorse that lasts. I hate the extruders that come on almost every Creality machine, and the hotend is limited to only 240 degrees. That said, \$200 is amazing, and I expect the price to reduce even further.

Belt (infinite Z) printing

This is something I had originally thought of as a novelty, but am really beginning to see the potential applications. Naomi Wu and Creality came out with what is known as the first commercially available belt printer with the Creality CR-30. There are now more options coming out from competitors as this is being written. I am actually reviewing the SainSmart Infi-20 as of editing this book, so that review should be out by the time you read this.

I have one of these CR-30 printers, which I reviewed on my YouTube channel, and it's pretty dang cool. Rather than the build plate moving back and forth as they do on Cartesian printers, this build plate is actually a belt – similar to a treadmill. On these printers, the build plate/treadmill is moving in the Z-direction, rather than the Y-Direction, which is why they are referred to as “infinite Z” printers.

The extruder then moves in the X and Y directions via a CoreXY setup, though it moves at a 45 degree angle to the build plate. The photo should explain this a bit clearer.



This setup allows for auto ejection of parts, since they are forced off as the treadmill curves around. This also allows for printing of “infinite” length 3D printed parts. Obviously not actually infinite, though you can print many times longer than the build platform actually allows for.

I printed a full sized skateboard deck as well as a 3 foot sword, with a printer that fits on my desk. The settings do require a bit of honing in, since the 45 degree angle seems to change what makes for the best quality print. Model orientation also needs to be factored in since parts at the front of your print will require more support structures than those on back of the print, and parts that didn't need support structures on a standard printer will need them on

these.

That said, once you hone everything in, this printer can run 24/7, auto ejecting parts and creating models larger than previously thought possible.

IDEX and Tool Changers

Unfortunately I have zero experience with tool changing printers, but they may very well be the future. The E3D tool changer allows for different extruders to be grabbed as they are required. This means that you can print in different colors or in different material types with entirely different extruders and hotends. The amount of tools or extruders you have is only limited by your particular setup. Your printer will grab an extruder, print what is required, park it, and then grab a different one.

This isn't only limited to 3D printing though, since you can have things that aren't extruders parked. This can also be used with laser engravers, CNC mills, sharpies, etc. The possibilities are endless and people continue to come up with unique ways to use their tool changer.

This tool changer is the best way to print with multiple material types, but the next best method would be IDEX, or "Independent Dual Extrusion," which I have tested on a couple of different machines. While more expensive than other dual extrusion methods, I have to say this is the only method (other than tool changers) to print multiple material types. I have tried to no avail getting dissolvable support material to work on all other methods, but my IDEX tests have worked without failure.

IDEX printers can actually print two of the same parts at the exact same time as well. This means if you have two prints that can each fit on half of your build plate, they will both finish in the time it takes to print one. This may be useful if you have to crank out a bunch of small parts in a quick time frame.

It is difficult for me to recommend an IDEX printer, unless you have a specific project that requires dissolvable support material or are doing the above example of printing two parts at the same time. That is what these machines are perfect for. I am also doing some composite testing by mixing both TPU and PLA together on one of these machines, hoping to result in a stronger part. My tests are not complete as of editing this book, but these types of prints would only be possible on an IDEX printer (or tool changer). If you do not need one of these particular applications, an IDEX might not be right for you.

Single nozzle, dual extrusion printing, which a lot of printers offer now, is nearly useless in my opinion. You cannot print multiple material types since you will inevitably encounter a clog, and they are really only good for two color printing. If you want to print in two distinct colors of one material, then those may be worth it for you, but I personally never have a print that I only want in two colors.

Failure detection

This is a burgeoning field that has me very excited. The company “Spaghetti Detective” was the first to introduce me to this, but essentially they utilize your Pi camera to see if your print is failing, and then pause the print. Their goal is to actually correct failures on the fly as the camera detects them.

Their current version requires you to have a Pi hooked up to your printer, along with a Pi camera, and then the software will send you a notification if it is detecting a failure, and will stop the print if the failure gets too bad. The main failure this detects is when parts are knocked off, and your printer is starting to create a “spaghetti monster”. This way you do not ever have to worry about coming back to a mess of a print with a ton of wasted material (and potentially a gunked up hotend).

Their goal over the coming years is to not only detect parts being knocked off, but actually be able to analyze what may be wrong with your print. If the camera detects over extrusion, it will reduce your flow rate. If the camera detects the nozzle is too far from the build plate, it will correct for that on the fly.

While these corrections aren’t quite live yet, I can see the potential in this over the coming years. In fact, if they get this product to work exactly as they want, things like my book may not be needed, since a self-correcting printer would be as plug and play as any other tool in your house.

The only other competitor to Spaghetti Detective that I know of is 3DQue, who are attempting to work on something similar to this. But as of editing this book, these can normally only detect major failures like parts being knocked off. Keep an eye on this technology though, since it may allow for printers to be more accessible to the general public.

Patents expiring

If you have followed my books, you know that there are a lot of annoying patents that have hindered 3D printing innovation. The expiration of these patents are one of the main reasons we have been flooded with inexpensive printers.

Well, some of the most useful patents have just expired, meaning we will soon see more affordable options in printing more unique materials. You may ask yourself why you cannot buy a printer that has an actively heated chamber – meaning it keeps the ambient air at a specific temperature. That is because it was illegal to sell these machines until very recently.

Fully enclosed 3d printers (not ones with random holes for the door handle and filament tubes) that are actively heated, should start flooding the market any day. As of editing this book, I have not personally seen an option that does this, though for all I know by the time this book hits your desk they will be commonplace. I think we will start to see that high temp, high strength materials are not only for industrial use. I would expect smaller, fully enclosed and heated printers to be coming in at around \$1,000 which will be very exciting to see.

Material Advancements

This is something that is continually advancing as new companies and current entities continue to come out with new material options. One of the most used that I didn't cover in the past is PLA + (or PolyMax PLA if using Polymaker.) Polymaker also is unveiling a new type of PLA that is exciting, which will be out by the time you read this book.

PLA+ has reinforced properties to have better mechanical properties over standard PLA, yet is just as easy to print. While PLA+ has a lower tensile strength than standard PLA, it has a much higher impact resistance, which is the thing that is most hindering PLA from being a usable material for mechanical applications. In fact, Polymaker's new PLA will have similar tensile strength as standard PLA, but will also have this impact resistance.

Because of this, PLA+ and PolyMax PLA have become easy to print replacements for many mechanical things. For instance, the 3D printed firearm community mostly uses this PLA+, which shocked me when I first learned about it.

The only issue is the lack of heat resistance, since PLA+ has the same glass transition temperature as standard PLA.

And then there are a near endless supply of new materials coming out just about every day. New combinations of materials, easier to print flexible materials, non-warping nylon, UV resistant ASA, and new dissolvable options are just a few that I can think of. I suggest following "Filament Frenzy" on Twitter since he is always trying out some cool material options.

Resin price reductions

This is a bit of an odd section, since these books have focused primarily on FDM 3D printing. The main reason for this was that I got into this hobby for its accessibility and the price of SLA printing was just inaccessible at that time. While FDM printing has come down in price, I think SLA has decreased even faster.

When I wrote the first edition of this book, I don't think it was possible to buy any SLA printer for under \$1,000, and that was the cheapest possible option. Now you can get a really good HD SLA printer with a build volume that is actually usable, for under \$200. The Elegoo Mars 2 is currently sitting at just \$199, and their Mars 3 is about to come out for just \$245, all with massive improvements in quality, print time, and usability. I currently have their larger Saturn SLA printer which works great and allows for some pretty large resin printing.

This means that if you are someone who wants to print miniatures, or just experiment with quality that isn't possible on FDM 3D printing, SLA printing is more accessible than ever.

Personally, I am not a big fan of SLA printing because it is very messy and requires space dedicated to potential spilling resin. This is because the way SLA printing is currently set up, it's pretty hard to remove a print and start a new one without getting a little resin dripped, especially if you need to swap out the material. I have frustrated myself many times spilling a droplet of resin on my carpeted office, so I really do not have the right setup for it.

All of that said, there is no chance I can FDM print something with nearly the same quality I can with SLA. Material options continue to expand as well, so don't write off SLA printing for your particular project needs.

Where improvement is still needed

While I just mentioned that price reduction and removal of patents have allowed for great and inexpensive 3D printers, it also means that some of the market has gone stale with hundreds of Ender 3 clones with slight modifications. The only printer, to this day, that I can honestly suggest as a machine that comes in at a reasonable price tag and requires no upgrades, is a Prusa. Lulzbot just came out with what they are trying as a competitor, though I personally think it is too expensive for what you get.

Many of you reading this book don't want to spend time upgrading your printer, though it is just about inevitable without spending over \$800. I don't think that needs to be the case, as there should be a lot more competitors to Prusa, and for a less expensive price tag.

You can buy an Ender 5 Pro or a CoreXY printer for under \$400, put a Hemera or BIQU H2 on it for \$110, and be rocking with a great printer. I don't see why a company can't do something similar with their own flair for under \$750, though I don't really know of one that exists. While I just said Prusa's are the only printer I can suggest that doesn't need any upgrades and works great, I still personally think it is a bit overpriced for what it is. That is solely based on how inexpensive all of these parts have become, though without any competitors, it's hard to say definitively that it's too expensive. Part of the reason Prusa's cost so much is due to their customer service, which shouldn't be overlooked. If you buy from Creality, good luck ever getting a response to an issue you have.

I think what would really help is for companies within the industry of making printer parts work more directly with printer manufacturers. One of the reasons Prusa's are so reliable is because they outsourced the parts to those who do it best. They use name brand Bondtech extruders and E3D Hotends. The same is true for what Lulzbot is doing. What is really holding Creality back is not working with companies like E3D, Bondtech, Slice, MicroSwiss and others to improve the components on their printers. Not off-brand clones or attempts to make it themselves, but work directly with those who do it best. That way they aren't the cheap printer that needs upgrades, but rather the perfectly priced workhorse that everyone will suggest to buy.

Hopefully 2022 changes this and we start to see some great all-in-one competitors.

Post-Processing

This is not going to be classified as a failure, but rather tips to help you combine, smooth, sand, and paint your 3D printed object. There will be no tips on how to print your part in this section, just how to post-process it.

I have had a decent amount of experience with post-processing, and will let you all know my results below.

Loctite Super Glue Gel

Combining 3D printed parts can be a difficult task, which is why Loctite Super Glue Gel has become the preferred favorite of many 3D printing enthusiasts. This stuff is quite strong, readily available, and can dry in under a minute. When you have a part that can't easily be clamped together for drying, or a small part that can easily break, you will likely want to go with using Loctite. Remember that this stuff dries fast so you can accidentally glue your finger to the part if you are not careful.

After you have used everything you can get out, make sure you remove the tube from the hard plastic bottle. There is still about 30% of the glue in there that you can't get out until removed from the casing.

This works great on PLA and ABS, as well as many other plastics, but it doesn't work great on everything. Nylon materials are very hard to find the correct adhesive.

Devcon Plastic Welder

This is a two part welder that you mix together and apply for parts you want to have a very strong hold. Devcon is just one manufacturer of Plastic Welders, I haven't tried them all. This is my go-to for any part I want to be as strong as possible. In fact, I have found that Devcon Plastic Welder is stronger than the actual layer adhesion, since on all of my tests the part would break before the bond did. The issue comes with the fact that it takes about 10 minutes to get a decent hold, and close to 24 hours before you get a fully cured weld.

Because of this, I use Loctite Super Glue Gel for those hard to hold and small spots, and Devcon Plastic Welder for just about everything else. If you can clamp your parts together easily, then plastic welding is likely the way to go.

The only issue I have found is that every so often I get a bottle where one part will come out at a different flow rate than the other. This has led me to wasting a decent amount of the product. I believe this is due to me being shipped old product, but it is hard to say exactly why.

Notes about Super Glue and Plastic Welding

These two products work great with PLA and many other hard plastics, but they do NOT work well with Nylons. I have only done a minimal amount of testing on bonding nylon parts together and I have yet to find something that I find acceptable. If you are designing a cosplay or fan art piece, use PLA or other hard plastics. If you are designing a mechanical piece to be printed in two parts in nylon, you should design those two parts screw or clamp together. Do not trust a super glue or plastic welding hold for parts that will be used in mechanical applications.

JB Weld

I have only started using this stuff recently since it is what is called on to use when building many 3D printed firearms, including the popular FGC-9. This stuff is pretty phenomenal, and can allow you to weld plastic parts to metal parts, and they hold together very well. So well that I couldn't get them apart no matter how hard I tried. You need to use a Dremel or even a soldering iron to melt the plastic off the metal you welded it to, otherwise the bond is pretty much permanent.

The process is similar to plastic welds, you have two parts you mix together. You get a hold within an hour and it fully cures in 12-24 hours. If you need a plastic part to stick to a metal one, and you want the hold permanent, check out JB Weld Steel. If it is good enough for 3D printed firearms, it is likely good enough for your project.

Starbond adhesives

Starbond has a wide variety of options and I have tested their thin, medium, and thick options, along with their accelerator spray. I prefer these methods now over Loctite, though they aren't quite as common and may be a bit more expensive. Personally I find the "thin" option a bit too viscous, since it is pretty much like water. It will get all over the place and really is only good for small cracks you want filled, since it will fall into any thin crevice you can think of.

The "medium" texture is likely my favorite, since it can go into cracks with ease but doesn't drip like water. That said, it is still a bit thinner than something like the Loctite mentioned earlier. So it is great for filling holes but might not be best for large parts you want held together.

For that, I go with their "thick" option. This stuff has a very similar viscosity to Loctite Super Glue Gel, and is perfect for holding parts together, though a little too thick for filling small gaps.

Their accelerator spray really does speed up the curing process, to the point parts stick together literally within seconds, but it does seem to add a white color to whatever print I spray. For this reason I only use it on parts I want to stick together very fast, and I do not care about the color/look of the print.

Other adhesives

There are actually quite a lot of adhesives on the market now specifically meant for 3D printing. This includes 3D Gloop, which has nothing but high praise from the community. I actually have never tested their products, but they have Gloop meant specific for PLA, one for ABS, and one for PETG. I doubt the community is wrong in their reviews, so if you want to test them out, I wouldn't tell you otherwise. This is particularly true for bonding two PETG materials together, since standard super glue doesn't work great on that.

And then the list of other possible adhesives is endless. Check reviews and what others say since many of these may be better than ones I have personally tested.

Sanding

Sanding is key regardless of the methods you choose to go with below, and will help you to clean up any print. For the majority of materials (including PLA) I start with 220 grit sandpaper. This is just about the right roughness to help smooth your print out without deforming the look. You can go slightly lower than this for very hard plastics, just be careful, especially if the material you are using has a low heat resistance. For ABS, you can actually start at a higher grit since it sands much easier than PLA.

From 220 you can move to 800 and even 2000 grit sandpaper. If you start with these high grit sandpapers, you won't get much progress, so it is smart to start with 220 and work your way up. You will likely want to wet the sandpaper as well since it helps in the process.

I use a circular power sander for many flat prints – especially those that I combine two parts, but you need to be very careful to not keep the sander in one spot for too long. This is because it will heat up your part past the glass transition temperature (especially with PLA), and can deform your print.

For hard to reach areas you can use a Dremel with a sanding tip, though once again you need to be very careful to only sand and not damage your part.

Bondo

This stuff is very hard to sand without damaging your part, so make sure you are only using it to seal large seams that need to be strong. I have used this a few times in the past and you can really get a smooth seal, it just comes with quite a lot of work.

When Bondo hardens it cannot be easily hand sanded. This means you will want to have an electrical sander (or a Dremel sander for small areas). This issue with this is that you will have to be very careful to not accidentally damage the printed part. Not only can you accidentally dent the printed part, sanding gets hot, so hot it can deform a PLA part.

But when it comes to having a strong seal where no one can see the seam, you will want to use Bondo. It just might be near impossible to perfectly sand a 90 degree angle. Otherwise you might want to try out Spackle.

Spackle

This is really for display pieces only, since it can accidentally be dented or scratched off with abrasion, even when fully dried. I only use this on small models that I don't plan on handling or using for mechanical purposes. For very small seams, or even small gaps in layers, I will rub on some spackle using my fingers. I am able to get a very small amount of this stuff into areas, and then after a half hour use some 800+ grit sandpaper to clean it all up.

The problem with spackle is that even after it dries it won't be that hard. This means you can accidentally dig a finger nail into it. This is why this is best for very small gaps on display pieces and should never be used on anything that needs to be used mechanically.

Model Putty

Model putty is kind of the perfect in-between of the two above, and is a bit closer to Bondo than Spackle. This means you will likely want to try out model putty if you want a strong seam but don't want to spend an immense amount of time sanding. That said, it is a lot more sanding than just normal Spackle.

Acetone vapor bath for ABS and ASA

One of the best parts of printing in ABS or ASA is the ability to acetone vapor finish your parts. PLA and other materials are not soluble in acetone, making them unable to be post processed this way. Not only can it make a print more water tight, it also gives a great finishing shine to parts that resemble an injection mold quality. Prints are smoother to the touch and overall easier to work with after acetone vapor finishing.

I have only started acetone vapor finishing parts again recently, since I now have good enclosed printers capable of printing large ABS and ASA parts.

Along with a high glass transition temperature, this is another reason you would still use ABS in 3D printing today.

You do have to keep in mind though that this can definitely be overdone and lead to a destroyed print, so use caution when proceeding with these steps.

*****CAUTION***** Acetone is **EXTREMELY** flammable, and this process should only be done in a well-ventilated area with absolutely no open flames. If you are not sure of your setup, DO NOT do it. In fact, there are non-heated methods of acetone vapor finishing which I highly suggest checking out first. Two such videos are titled “Improve your prints with acetone smoothing” by Prusa 3D, and “How and How NOT To Do Acetone Vapor” by Let’s print. This process takes much longer than the one I describe below, but is also highly preferred to be safe. Acetone vapor is ridiculously flammable and you will have a very bad fire if you are not careful.

If you want to do it the way I do, I have a video describing the method below titled “Smooth your ABS and ASA Parts with Acetone”.

Step 1: I used to use a slow cooker for this, but I now just use a standard cooking pot and my heated build plate. You will want to either use a metal grate, or even better, hang your prints via some fishing line. No matter what you do, whatever your print is touching will get a small amount of deformity. This is why I prefer hanging my part from the lid of my pot via fishing line, as to reduce these marks as much as possible.

Step 2: Place your pot with your part hanging or on a grate onto your heated build plate. I personally heat my build plate to 65-75 degrees Celsius. This means you will have to keep an eye on your part, since this acetone will vaporize pretty fast. Once your pot reaches the build plates temperature, it will only take a couple of minutes. Once you see evaporation on the inside of your lid, it is time to remove the pot. You can always do multiple passes but you cannot save a part that has been deformed.

I actually used to use a big broiler with a similar setup that I would put onto a hot plate for larger ABS prints. This process requires roughly 10 minutes though, due to the large volume of the broiler. That said, if you are acetone vapor finishing a large part, you will need some ventilation inside your pot. This is because the acetone vapor is heavy and will reside only at the bottom. For it to reach the top of your print it will need a small fan installed to your chamber to move the air around.

I recommend that you always error on the side of less time. You can always do another round of acetone vapor if you determine that the print requires it.

Step 3: After your quick acetone vapor bath is complete, remove the grate that is holding your part and put it off to the side (onto a countertop that you do not care about). If you are hanging your print via fishing line, find a way to rest your lid onto something that allows the part to hang to dry. I actually just stack a couple of spool boxes on two sides for the lid to rest on, while the part hangs in-between them. My video on the topic shows examples of this. After allowing for 30 minutes or more for drying, you can remove the part to be vacuum purged.

This step is not needed but definitely helps with the strength of the part as well as the time required to dry. Doing the process above without a vacuum purge will require 24 – 48 hours before the part is to full strength.

Note: Now that I have parts which were acetone vapor finished over two years ago, I am noticing some issues with this process. While those parts looked great for a long time, they are now showing some cracks. I can't explain exactly why this happens, but feel it necessary to let you know that it has happened to my acetone vapor prints. Perhaps they just had too much acetone vapor finishing, though I cannot say for sure.

XTC 3D for PLA and PETG

I like to be entirely honest with my reviews, and while other's have seen some amazing results from this stuff, I just personally have not. I am not sure if I am doing something wrong with the mixing or what, but my results are never quite what I see other's getting. Because of this, I haven't actually done this to a PLA print in about 4 years.

The idea behind this is XTC is that it is a protective and smoothing coat for finishing 3D parts that does not melt plastic. XTC-3D should fill in gaps and retain a smooth, shiny finish.

I have definitely had some semi-successful results in the past, it just often becomes more work than it is worth. First, you have to make sure the part is sanded as best as you can with standard 220 grit sandpaper. You then need to wear gloves and be in a well-ventilated area as you mix the two part goop. You then paint on an even coat, wait for it to entirely dry, and give it a further couple passes of sanding. After a decent amount of elbow grease, you should be left with a very smooth outer surface.

With some practice I am sure this can be perfected, I just personally prefer other methods at this point and no longer suggest it.

Polysher and Polysmooth

Polymaker has also made a product called the Polysher which works exactly as an acetone vapor bath, but is made specifically for 3D prints utilizing isopropyl alcohol. Rather than being soluble in acetone, their proprietary PolySmooth PVB filament is soluble in alcohol, making a vapor bath work perfectly to remove layer lines.

I HIGHLY recommend going this method for any part you do not need to be mechanically strong or heat resistant over acetone vapor and ABS. If you just want cosmetics, do not go with the more dangerous route of acetone vapor finishing, and just use Polysmooth or other PVB type material with an isopropyl alcohol bath. It works just the same, you will just not have as mechanically strong of a part.

I was given a Polysher for review a couple of years ago (which is up on my YouTube channel) and was very happy with the results. It is much safer than going with an acetone vapor bath, but you are limited to only using their PolySmooth material (or another alcohol soluble filament), and you are also limited on size to their Polysher.

That said, you don't actually need their Polysher to do this. You can use the exact method I explained for acetone vapor finishing, just with using isopropyl alcohol instead. In fact, you can even just spray the alcohol onto the print with a mister or spray bottle, though obviously using the Polysher is preferred.

This is great if you plan on doing a lot of display pieces such as miniatures or action figures, since it works perfectly to result in an injection mold look - but I would be cautious when working with mechanical parts due to the lack of detailed information on PolySmooth. It also does not have a very high heat resistance.

For further information, search for "Polysher by Polymaker Review" on YouTube for my 3D Print General review.

The spraying method does make it so the underside of your print will stick and deform if you are not extremely careful. You can do multiple passes of sprays to get even smoother results, you just really need to wait a good 24 hours in-between sprays, with a good fan focused on your part during that dry time. If you have a way to heat the part up a bit and perform a vacuum purge, this time will be decreased a lot.

What is pretty amazing is their PolySmooth clear material can also use this process. With a few coats of sprays, I am able to get this material just about entirely transparent, so long as it is only one shell wall thick without any

infill. I have never been able to get something quite as clear as this. You can see how it is done by watching my two videos on the topic called “Transparent 3D Prints” and “More Transparent 3D Print Tests”.

Primer

If you plan on painting your part, you will want to spray a coat of flat grey primer (or primer filler) before moving forward. Make sure you have combined your parts, sanded them smooth, and cleaned them entirely off of debris before going to this step. It would be smart to use a powerful air blower if you have a compressor on hand as to make sure all dust is blown off.

In a well-ventilated or outside area, with a tarp laid down, spray a light, even coat of primer about 6-12 inches from your part. This will allow for acrylics and other paints to stick to your part properly.

After waiting a couple of hours to fully dry, you can go ahead with spray painting or hand painting. If you went ahead and used primer filler, you can actually further sand smooth your print, which is why I prefer it over standard primer. When I want a really smooth print I actually do a coat of primer filler, wait to dry, sand, clean off, spray another coat of primer filler, and then sand once more. You may lose some fine detail when going this route, but layer lines should be gone.

Painting

Over the past couple of years I have done some testing with using an airbrush on 3D prints, and have slowly gotten better at it. You can see my first attempts at learning how to do this in the video titled “Learning to Airbrush 3D Prints”, but many of my videos after this I use air brushing as well.

I have found it very difficult to paint within lines using an airbrush, but it works amazingly to get an even spread of paint, as well as for shading. If I want to cover an area in paint but want it as smooth as possible, I go with airbrushing. Hand painting just leaves too many brush marks or clumps. If I want to get into shading, I also use airbrushing. You can see how you can improve prints by watching the videos titled “3D Printed Stan Lee”, “Captain America with Mjolnir – Print and Paint”, “Lifesized 3D Printed Nefertiti – Scan the World” and many, many others under my playlist “Print Everything!”.

If I need to paint within a small area, I will always go with hand painting. Since I was hand painting my prints for a couple of years before purchasing an airbrush kit, I have a bit more experience in this regard.

You can get a 24 color set of acrylic paints online for pretty cheap, it just requires some practice to get details down. You can also use model paint, it is just a tad more expensive. I have a few tutorial videos in this regard on my YouTube channel if interested in learning further. Remember to always allow time to dry and finish with either a clear satin or glossy spray coat to make sure everything sets as it should.

I suggest getting a thin set of good paint brushes to make sure you can stay within the lines. I always go with hand painting for features such as eyes due to the fine detail required.

Keep in mind that not all materials are great to paint. I have attempted to paint flexible filaments in the past, and it seems that acrylic paints will crack when being flexed.

I have also learned how to paint eyes properly from a great tutorial video titled “GalactiCustoms: 1/6 Paint Tutorial: Obi-Wan Kenobi- Pt 3 Eyes”. The title is a bit long, but his description on how to paint eyes have transformed the look of my painted parts.

Using shoe polish for shading

This is something I don't really use anymore ever since getting an airbrush kit, but shoe polish can also add a great shading effect to parts that call for it. You need to make sure your part is as smooth as possible, the thicker the gaps in the layer lines, the worse this effect will be. You also need to make sure the previous painting is entirely dry with a clear coat on top, as to reduce any chance of paint chipping.

Essentially, if you get some black shoe polish and spread it out on a paper plate, you can then brush it onto your print with a paintbrush into the crevices and indents that should show shading. This works great for detailed figurines in which you want muscles or folds in shirts to show through better.

The black shoe polish will naturally go into these indents, and all you have left is to clean off the excess. Grab a sponge and get it wet. I used to suggest doing this after the shoe polish had dried, but I have found it easier to just wipe off sections as you go. Attempting to brush off dried shoe polish will require enough pressure that you may chip off paint.

If you have large layer gaps and haven't sanded your part smooth, I would suggest avoiding this method, since this black shoe polish will go into these layer gaps and make them more noticeable. This is best for prints you have smoothed via sanding or vapor smoothing.

For a far more detailed explanation, you should check out Cosplay Chris on YouTube, as he has perfected this approach with his "Custom Collectables" playlist.

Upgrades & Purchasing a New Printer

3D printing technology is rapidly improving and even purchasing this book just two years from publication can have many of these failures fixed on stock new machines.

These are parts that you may want to think about upgrading on your machine, or what you should look for when purchasing a new printer.

Upgrade hotend

Many high end printers come stock with a well-made hotend, but the vast majority of inexpensive machines do not. If you bought a Creality machine, you won't have an all-metal hotend, and you are limited to printing at a max of 240°C without upgrading. I think this is the most important part to upgrade if you want to play around with materials other than PLA, PETG, and TPU. Though ABS can print at 240°, I actually suggest going a bit hotter than this, which wouldn't be reliable without an all-metal hotend.

If you read somewhere in this book to extrude at a temperature above 240°C, make sure you have an all-metal hotend before continuing with the instructions. All-metal hotends do not need a PTFE tubing going all the way to the heat break, meaning you can print much hotter materials.

I have personally standardized to E3D hotends, and it seems like a lot of the 3D printing community has as well. They are well made, all-metal, and have a variety of options. From the extremely popular and well-made Prusa MK3S+ all the way to the expensive Raise3D machines all use E3D hotends. That said, there are now competitors that are definitely stealing market share, one of the biggest being Slice Engineering and their Mosquito hotends. Unfortunately I have not used one, but they have a lot of love in the community and are worth looking into as an alternative to E3D.

If you are using a Creality machine and you want the easiest swap possible, then you should look at MicroSwiss. MicroSwiss makes drop in ready hotends for Creality machines, meaning you won't need to print anything and swapping your hotend is as easy as swapping some wires. They work just fine and I have no complaints about MicroSwiss, I just have only used one in my printing history.

E3D has a wide array of nozzle diameter choices, as do most competitors, and they are well-made and seem to experience minimal amount heat creep. And if you do experience a nozzle clog, they are easy to disassemble. You can see all of these options over at [MatterHackers.com](https://matterhackers.com).

If you find a different hotend that you prefer, you should definitely go for that. Just make sure you watch any relevant reviews first and be extremely cautious of attempting to save money from off-brand versions. You generally get what you pay for when it comes to knockoffs.

Replacing a poorly made or very old hotend can drastically reduce failures, increase the amount of materials you are able to print, and improve the quality of your parts.

Geared extruder

A stock direct extruder without any added gear ratio cannot print nearly as fast as a well-made geared extruder with dual drive. You can print an extruder upgrade, but this isn't very common anymore now that there are so many good extruders on the market.

An extruder without a gear ratio just means that each turn the stepper motor makes is directly related to how the hobbed bolt/gear turns. For example, every Creality machine comes with a non-geared extruder. Having a gear ratio means that the extruder stepper motor has to turn faster to extrude the same amount of material, but it also means it requires less torque and will result in far fewer extruder motor skip due to the mechanical advantage. It will also allow for a wider range of printing options.

Without a gear ratio you will be limited to printing much slower than printing with a gear ratio, and you will not be able to print as flexible of filaments. The same is true with nozzle diameters. Attempting to print on a nozzle with a diameter smaller than 0.3mm will almost certainly require an extruder with a gear ratio. Both of these facts are even truer when using a Bowden printer instead of a direct version. While Bowden setups have the benefit of reduced weight on your carriage, they almost always result in less printing options.

One of the most popular extruders as of my previous editions was the Titan by E3D. That is a well-made extruder with a gear ratio. That said, there have been a few upgrades over the years.

Bondtech came out with the first commercially available dual geared extruder that I know of. Dual gear, or dual drive, means that it has the filament being gripped and fed on both sides, rather than just one pushing against a bearing.

These extruders became a favorite in the 3D printing community, though I do suggest reading "Patterns in Outer Surface" chapter to see their one drawback as of right now. I haven't experienced the issue explained in that chapter, but it is worth mentioning if you are thinking of upgrading.

Since this Bondtech BMG came out years ago, the next big advancement seems to be combining this gear ratio dual drive extruder with the hotend, as to reduce any gap between the extruder and heaterblock.

The first company to do this was E3D with their Hemera extruder. I bought it right when it came out, and it instantly became my favorite extruder/hotend combo.



The gap between the gears feeding filament and the heaterblock is almost non-existent, and this makes printing flexible materials as easy as printing PLA. I have tested printing Ninjaflex, one of the hardest flexible materials to print, and was able to get a successful print at 100mm/s. I was lucky to get a successful print at 25mm/s with Ninjaflex before the Hemera, so clearly these upgrades are working.

The one thing holding back the Hemera in my opinion is the size and weight of the stepper motor. If this concerns you at all, I would suggest looking at the BIQU H2 or the OmniaDrop by DropEffect, both of which use lighter pancake stepper motors. Keep in mind that BIQU does not have the same quality control that E3D has, since even my review unit had setscrews loose that shouldn't have been. I have heard similar complaints from people who bought the H2. DropEffect is a small company but seems to have good customer support and quality control from my experience.

That said, after I tightened everything up, the H2 is a great competitor to the Hemera, as is the OmniaDrop. They accomplish the same tasks as the Hemera but at a much lighter weight. I personally have my H2 on an Ender 5 and it has become one of my go-to machines when I want a successful print. While this H2 a great little extruder at a really great price, just keep in mind you might not get the customer support and quality control as you would with E3D.

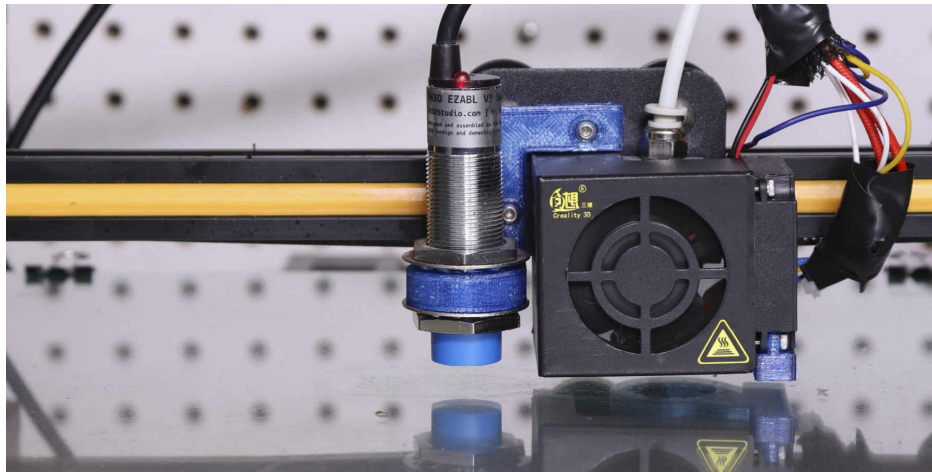
I personally have not tested a single extruder from Bondtech since the BMG, but they have also made improvements to that design. Their competitor to the Hemera is called the LGX, which is a dual drive extruder/hotend combo, but with a slice engineering hotend rather than E3D. It really looks interesting and something I want to play around with. It is definitely worth looking at as an alternative to the Hemera, OmniaDrop and H2.

Auto bed leveling/auto bed tramming

Five+ years ago, the vast majority of manufacturers that offered an auto bed leveling system did not live up to their marketing. They were too mechanical and overall did not work much better than just adding a few extra end stops.

By 2018 you could purchase 3rd party auto bed leveling lasers, or buy a machine that comes stock with one. Many of these work quite well and can save you the endless amount of headaches that I describe in the “Unlevelled Build Plate” chapter.

Bed levelling is technically bed tramming, since the printer isn’t actually levelling the bed. Rather it is taking a mesh of your build plate to determine how uneven it is, and then it changes the Z-height of your print as it travels to compensate. Because of this, it is still suggested to get your build plate as level as possible even if using a bed leveler.



Pictured above is one of the first versions of the EZABL by TH3D Studios, which worked great as advertised. They have since done upgrades to newer models, though it seems the most popular bed leveler has become the BL Touch. The BL Touch is a mechanical bed sensor rather than the EZABL optical sensor.

Surprisingly, even inexpensive machines are now adding these. I have a \$200 Tina2 sent to me by WEEDO for review which actually has one of these stock as well. Upgrading your Ender 3 to have a BL Touch is less than \$50, and Creality has made it really easy to add one. I would never have expected this to be the case a 4 years ago when I released the first edition of this book. If you are spending over \$500 on your machine, it should hopefully come with a bed leveler stock.

These sensors will be mounted to your carriage, and you can find a model to mount to your printer setup on Thingiverse. You should watch videos on

TH3D's YouTube channel, since Timothy covers a lot of issues and setup guides for adding this to your machine. I also have a quick video where I go over this on my YouTube channel titled "Paid Upgrades for the CR-10", though the process should be even easier now on modern machines.

To be honest, I have come to not really care if my printer has a bed leveler anymore. Over the years it has become second nature to me to visually see if a build plate isn't level. I can easily screw or unscrew corners of the build plate on the first layer if I see something isn't right. That said – an unlevelled build plate can be the biggest nuisance for someone new to 3D printing, so these devices have really made 3D printing more accessible.

Get thick Z-axis leadscrews

A great improvement to think of when buying your next printer is to see how thick the threaded Z-axis rod/rods are. The thicker the rod and the better it is housed into the frame, the longer your printer will last without having tolerance issues.

Thick M10 threaded Z-axis leadscrews on a printer usually means the printer will come with a higher price tag, but it is definitely a great upgrade. Many well-built CoreXY machines have a single thick M10 leadscrew which the build plate moves up and down on. Inexpensive Cartesian machines will have a single M5 threaded rod, while better versions will have two M8 leadscrews. My main workhorse printers are CoreXY and they have two M8 leadscrews.

The standard Ender 3 comes with only one leadscrew, and it will likely work fine with just one due to its smaller build size. That said – it is better to have two leadscrews for precision and longevity of the printer.

Cartesian vs. CoreXY vs. Delta

There are benefits to each of these types of printers, but the vast majority of machines you will see on the market are Cartesian, with CoreXY being the second most common.

Delta printers seem to be able to print the fastest based on how the movements are done, but can be difficult to calibrate and must be tall if you want to print wide objects. They are also much harder to print in a direct fashion, since almost all Delta printers are Bowden. I have played around with a few different FLSUN Delta printers, as you can find on my YouTube channel, and their SuperRacer is a pretty good machine with a great build quality. I just personally do not like Delta's due to their height and lack of direct extrusion.

CoreXY printers have been growing in popularity and they definitely have added sturdiness and benefits. I personally like CoreXY machines the most for quality, but have a bit more experience on Cartesian models, due to them being more popular over the past decade.

CoreXY machines have the benefit of not moving the build plate back and forth. This constant moving of the build plate will not only have extra wear and tear on your printer, but can result in parts getting knocked over easier with an increased chance of ghosting.

Because of this, CoreXY machines have become my favorite type of printer. That said – printers like the Ender 5 are technically Cartesian in the motor movements, though it also has the build plate moving up and down, meaning it has many of the benefits that CoreXY machines offer.

In fact, there are new machines like the Voron builds that don't have the build plate move at all, where the hotend moves around in a CoreXY fashion and the entire carriage is moved up and down in the Z-direction. Generally the less movement the build plate is doing, the better.

Enclosed machine

I go over this added benefit in the “Warping” chapter in this book, but it is highly beneficial to purchase a machine that has its build area at least somewhat enclosed when printing with warping materials. This way you do not have to worry as much when wanting to print in materials that have a high internal stress. You just need to make sure your board is not enclosed with the printer, since you don’t want that overheating.

I now personally have a few “enclosed” printers, though none are technically fully enclosed due to patents that have recently expired. These make printing high warping much easier. You just would want to make sure and remove the top, or open the door, if printing in PLA - since you don’t want the ambient air getting to its glass transition temperature.

Metal frame

I don't think this comment is really needed anymore, since it is extremely rare to find an acrylic framed printer today. That said – a sturdy metal frame is really needed for a printer that lasts. Stay away from old acrylic or wood cut frames.

Linear rails

3D printers on the market are increasingly offering linear rails for their frame instead of linear rods or aluminum extrusions. Linear rails use a stiff, steel rail along which the carriages slide via bearings. This is different than a linear rod printer where the carriages are attached to a smooth rod via bearings. As an example, the Prusa machines use linear rods.

The vast majority of printers a couple of years ago used linear rods, but it seems to be changing. This is due to an increasing amount of companies making printers using an aluminum extrusion frame, such as the popular Creality machines. These have the carriages move along aluminum extrusion bars via rollers. This has become one of the most preferred way to make a printer, likely due to the cost being less to produce.

Some people prefer linear rods to aluminum extrusion, though I personally prefer the latter. A well-built linear rod printer such as the Prusa should be better than aluminum extrusion, I have just had quite a lot of linear rod printers using subpar bearings or attaching them poorly, which can cause free play.

That said, I believe that just about every maker prefers linear rails. They are sturdier and have very smooth movements, they just generally cost more. All of the super-fast Voron builds utilize linear rails.

24V instead of 12V

Most inexpensive 3D printers used come with a 12V power supply and output, though this seems to have changed over the last couple of years. Almost every printer I reviewed a couple of years ago were 12V, but now it seems they are all 24V.

Buying a printer that is 24V will come with a few added benefits. These include a shorter time required to heat your build plate, more torque to your stepper motors (reduces extruder stepper skipping and allows for faster prints), and results in less noise produced by your stepper motors.

If you are building a printer from scratch, or are upgrading your current machine, just make sure any upgrades you purchase are rated properly. If you are converting your printer to 24V, you will need to change your fans, hot end heater cartridge, your heated build plate (or at least the wiring to it if your build plate can handle 24v), and you must remove a specific diode on a RAMPS board. If you are doing this upgrade yourself, you must watch tutorials and be sure you are confident in what you are doing. If not confident, don't do it.

Keep in mind a 24V machine is likely going to be more expensive than their 12V counterparts, though that is not always the case. Do some research on the particular machine you plan on purchasing and if it being 24V is beneficial to you.

1.75mm vs. 3.00mm filament

This debate has been going on for years and there is still no specific answer, though it seems that 1.75mm has slowly won. I have extensive experience using both, and personally I like to use 1.75mm more.

During the first edition of my book, I would say that companies were split on which to focus on. That said, if you are new to 3D printing, it is likely you will not ever encounter a 3mm filament printer.

3.00mm filament (more accurately it is 2.85mm filament) should have a tighter tolerance in its diameter than 1.75mm filament does. This should theoretically mean parts should come out cleaner and you should experience less nozzle clogs, but I do not notice much of a difference in this vs 1.75mm printers. You can print parts a bit faster though because less torque is required on your extruder stepper since less distance is required to extrude the same amount of material. This means your extruder can turn slower when dealing with 3.00mm filament (possibly leading to less extruder motor skips).

One of the biggest issue with 2.85mm filament is with its want to curl back onto its spool. When you are near the end of a 2.85mm PLA spool, you can experience an extensive amount of breaks in the filament due to its tendency to curl back into a circle.

This is far more annoying than you can imagine if you haven't dealt with it yourself. This factor alone has made me prefer 1.75mm filament, since it is far easier to manipulate and you don't ever have to throw away spools that still have 50 grams of material on them. I see no quality differences with 1.75mm and the headaches involved with using it are far less. You may prefer 2.85mm, I have just come to like 1.75mm, as it seems the majority of the community does as well.

Poll Results

I actually held a poll recently to see what the community prefers. I haven't tried every setup, so I wanted to see what everyone likes to use. Below are the results for a poll that had 189 responses, so it should be a decent gauge.

I suggest printer manufacturers refer to this in order to build a great factory machine, and for you to reference for your upgrades.

Build Type:

Cartesian with Build Plate moving in Y Axis: **37%**

Build Plate Moves in Z direction such as CoreXY: **50.3%**

Delta: **2.1%**

Other (such as Voron where build plate doesn't move): **10.6%**

Notes about above poll: I personally believe that majority of people would prefer the Voron style, it just often comes at a higher price tag. I think that the Cartesian option only had such a high result because it is so common, and many people have not tried these other build types.

Extruder Type:

Bowden: **21.3%**

Direct: **64.9%**

No Preference: **13.8%**

Frame Type:

Linear Rods (Prusa style): **8.6%**

Aluminum Extrusion with Rollers (Ender Style): **28.3%**

Linear Rails: **46.5%**

No Preference: **15%**

Notes about above poll: There were a couple manual entries (which is why this doesn't equal 100%) with unique combinations. Some like rods for the Z-axis, and rails for the X and Y movement. I also believe the roller option got so high of a result because there are just so many printers that use this setup, especially inexpensive ones. In general, linear rails are normally the best option, though this may be tweaked for the Z axis.

Filament Diameter:

1.75mm: **93.7%**

2.85mm: **2.1%**

No Preference: **4.2%**

Preferred Build Size:

200mm in each direction or less (smaller than Ender 3): **5.8%**

Roughly 250mm in each direction (Ender 3 size): **35.4%**

Roughly 300mm in each direction (CR-10 size): **35.4%**

200-300mm in X and Y but a short Z (Easier to stack): **3.2%**

Infinite belt with reasonable X and Y: **0.5%**

The larger the better (Over 300mm in each axis): **18.5%**

Other or unique response: **1%**

Favorite Extruder:

E3D Hemera: **14.2%**

E3D Titan or Aero: **8.9%**

BIQU H2: **3.6%**

Bondtech LGX: **5.9%**

Bondtech BMG or other Bondtech: **30.2%**

MicroSwiss: **17.8%**

Lulzbot Tool Heads: **0.6%**

Remote Direct Drive: **2.4%**

Orbiter: **3.6%**

No Preference: **9%**

Other: **3.8%**

Notes about above poll: It seems there are just so many extruder options on the market today, it is difficult for there to be a community consensus. It seems that Bondtech BMG's are still very popular even though there are new models. I am not sure if that is because that is what the community prefers, or if they just haven't had the opportunity to try the new models.

Favorite Hotend:

E3D V6: **26.6%**

E3D Volcano or SuperVolcano: **9.5%**

Slice Engineering Copperhead: **3%**

Slice Engineering Mosquito or Magnum Mosquito: **17.2%**

MicroSwiss: **20.7%**

Phaetus Dragon: **13.2%**

No Preference: **4.8%**

Other: **5%**

Notes about above poll: A lot of these hotends, including ones by Slice Engineering and the Dragon option, I have never tried. I didn't even know that the Phaetus Dragon existed, yet it scored pretty high on the results. This means it is worth it for you to research the hotend of your choice before just going ahead and purchasing an E3D one because I use it. I assume MicroSwiss scored high because it is very easy to drop in an all-metal version into your Creality printer without any need to cut wires or print a new mount.

Favorite Nozzle Diameter Size:

0.3mm or smaller: **2.7%**

0.4mm: **78.5%**

0.6mm: **14.5%**

0.8mm: **2.7%**

Larger than 0.8mm: **1.6%**

Preferred All-Around Build Plate Type:

Whatever came stock on my printer/No preference: **13.3%**

Glass with basic additives (hairspray, glue, etc): **29.3**

Wham Bam/TH3D/Fulament PEI flex plate: **38.8%**

Full PEI build plate (Not flex plate): **6.4%**

BuildTak Sheets or similar: **2.7%**

Magigoo line of adhesives on any build plate: **2.1%**

Other: **7.4%**

Favorite Slicer:

Cura: **64%**

PrusaSlicer: **24.3%**

IdeaMaker: **3.2%**

FlashPrint: **0.5%**

Slic3r: **1.1%**

Simplify 3D: **1.6%**

Super Slicer: **5.3%**

Notes about above poll: I am a bit shocked just how Simplify 3D has fallen in popularity, I would have expected more people still use it. But that just shows how these free slicers have gotten so much better by offering frequent updates. I had never heard of Super Slicer before this poll, so it is worth checking that out, along with PrusaSlicer and IdeaMaker if you do not like Cura or its user interface.

With so many options of printers and upgrades now available on the market, there really isn't a one size fits all choice. These poll results should at least give you, and hopefully printer manufacturers, a good starting point to find the best printer for you.

Resin Printing



Let me preface this section by saying that I am not a resin printing expert. I have only worked on 4 different SLA machines, all with LCD screens and all under a \$600 price point. I personally now only use the Elegoo Saturn, and have never used any FormLabs style laser machines. I am including this section as an introduction to someone brand new to resin printing who is looking for a similar machine setup.

If you have more than 100 successful prints with an SLA printer, you are more experienced than me, and can likely skip this section. But if you would like to get into this other style of printing, I figured I would go over some learning experiences that I have encountered. Since I am no expert, I won't be going into specific details about types and styles of printers, and will only talk about what I have personally used.

Things they don't advertise

While resin printing has some of the best quality of 3D printing I have seen, far superior to FDM 3D printing, it seems that no resin printing company advertises just how much cleanup is involved. Every time I am done resin printing, it takes nearly an hour just to clean the vat and entire setup.

You will want your resin printer in a section of your workshop that is OK with getting a bit dirty. My office is carpeted, and I intentionally avoid resin printing because of this.

No matter how much you try, it is inevitable that you will have a bit of liquid resin or cured resin flakes that end up where you don't want it. Make sure you have a large silicone mat, and I would also suggest adding contact paper to whatever work bench you decide to scrape prints off onto.

And that is assuming you do everything properly. I have accidentally spilled some resin, and not only can it be toxic, it will not be easy to clean up, and will certainly ruin your carpet if you have any. This resin can also smell, so you will want good ventilation.

For this reason, I only use resin printing if I am doing a particular video that requires it, or if I have a lot of prints lined up in a row. You don't want resin sitting in your vat for long periods of time, so you really should clean everything up without the resin sitting still for more than 24 hours. You can leave the resin in the vat for a couple of days, so long as there is no UV light hitting it, but you will definitely want to make sure to mix it before you start printing again. I recommend no longer than a day to be safe, but you can definitely leave it for a few days. So if you just have one miniature to print, then cleanup alone may take longer than the print itself.

I don't intend to turn you off from trying resin printing, because as I mentioned the quality cannot be matched. It is just something you should keep in mind before you buy something and later realize the level of cleanup that is required.

An important upgrade

To avoid even further annoyance with removing prints, a very smart upgrade would be to get a flex plate. These flex plates are made by Wham Bam, Fulament and others, and trust me when I say they will save you a lot of headaches.

Removing a print from the buildplate with resin printing has proven a lot more difficult for me than with FDM printing. Not only do you have to be careful not to make a giant mess on your work table, some prints are near impossible to remove. This is one reason it is always suggested to lift prints off your build plate, or print them at an angle, since a large flat surface can be so difficult to remove that you may damage them.

I have hacked at prints on the build plate, only to have them flung across my room, leaving resin all over in its wake. These flex plates work very similarly to FDM printing, but they are far more important in resin printing – at least in my opinion.

A simple bend of the flex plate will remove these parts that previously required a lot of force to remove.

Be prepared

As mentioned, you will want a good work station to remove and clean up resin prints. A large silicone mat and contact paper are necessary. Then you will want to make sure to have plenty of nitrile gloves – make sure they are not latex. These can be purchased in bulk for cheap on Amazon.

You do not want any resin in contact with your skin. This means you will go through a lot of gloves. I wear a pair whenever I pour new resin into the vat, which then means I will need a second pair when it comes time to clean the print off. This is for every print, so I will go through at least one pair of gloves for every single build plate that I print.

Resin also cures via UV light, which is how this printing process is done in the first place, so you do not want any sunlight or UV light hitting your resin vat. You can print a cover for your resin vat when it is not in use, though as mentioned, you don't want resin just sitting in there for multiple days or weeks at a time.

This means I always make sure my lid is on my resin printer when working in my office, since I have very large windows that let in sunlight. This isn't as important if you don't have windows like mine, but regardless it is always safe to try and prevent as much UV light as possible from hitting your vat.

Have plenty of isopropyl alcohol. This stuff was hard to come by during the peak of the pandemic, so I ended up using denatured alcohol, but you will go through quite a lot of this stuff. If you are printing larger prints, and not just miniatures, you will be surprised just how many bottles you go through. This isopropyl alcohol is also good to clean up your vat when you are done printing, as well as all of your tools that get resin on them such as scrapers.

The FEP film that is on the underside of your vat is a consumable product. These FEP films will get damaged and will inevitably need replacing – sometimes faster than you would assume. These are inexpensive, though it is worth mentioning that since many printers do not come with extra, be sure to purchase extra FEP films sized for your printer so that you have them for when yours inevitably gets damaged.

Slicing for resin printing

As mentioned before, I am not an expert, so I usually just follow the exact settings I am told to use by other creators. My favorite YouTuber in this regard is “3DPrintingPro”. He has quite a few ChiTuBox tutorials, which is the main slicer that you will use when Resin printing with machines such as those made by Elegoo.

There is some recent controversy with ChiTuBox creating a “Pro” version that costs money, but the standard version should work fine at the time of writing this book, and it is something you will have to become familiar with.

Whenever I print with a new printer, setup has been simple on ChiTuBox, and their starting settings seem to work fine for me. Some notes are that the curing time can change depending on your printer and the resin you are using. Monochrome printers let in more UV light than standard LCD, which means it will cure faster. Since we are curing via UV light, transparent resins will cure faster than opaque, sometimes twice as fast.

When using my monochrome 4K Elegoo Saturn, my individual transparent layers when at 50 micron layer heights cure in about 3 seconds. But when I use opaque black resin, I change that to 6 seconds. This means the print will take just about twice as long, but it is important to make sure opaque resin cures properly. The thicker the layer height, the more you want to consider increasing the curing time. Remember this is specific to LCD resin printing, since laser resin printing, such as those made by FormLabs, is a different process and takes a lot longer to cure. LCD printing will cure the entire layer at once.

When it comes to supports I just follow the exact settings that 3DPrintingPro has on his video titled “I’ve updated my insane support settings for Chitubox!!! A quick video walkthrough of my new settings.” I then manually add heavy supports to sections that are floating in the air, medium supports to sections that are an extreme angle, and light supports to small sections to make sure they come out clean.

This addition of supports usually takes me a little while, since I don’t trust adding auto supports. Auto supports have often either damaged the undersides of my prints, or they don’t offer enough support. 3DPrintingPro’s settings seem to work great for me.

This then leads to print orientation. Print orientation is a bit different than FDM printing, since you don’t want a giant flat surface touching your build plate. So with resin printing you will often want to angle your print so only a small surface area is on the build plate, or do what I do and raise your print

entirely off the build plate with a “Z Lift Height”. I usually do 5mm off the build plate and then add heavy supports to the undersides that are floating in the air. Trust me when I say having a very large surface area touching the build plate will be very hard to remove.

One other thing to note about slicing for resin printing is that you will be limited to either printing a part entirely solid or hollow. There doesn't seem to be a way that I know of to have a specific percentage infill. This means that the vast majority of my prints are 100% solid, and thus use a lot more material than FDM printing. If you decide to print your part hollow, you will need to add holes at the top of your print (or bottom if looking at your printer) so that liquid resin can escape. You can increase the wall thickness on these hollow prints, since too thin can actually warp the walls. Remember overhangs will still be an issue if you print hollow, and you will be far more likely to have resin trapped inside your print, so holes will be needed.

Methods to clean and cure prints

Before scraping your print off, or popping it off with a flex plate, you will want to have a vat large enough to hold your print filled with isopropyl alcohol. As mentioned before, you can use denatured alcohol as well if required. I personally have two vats, one for initial cleanup and one for further cleaning, since the initial vat will be full of resin after just a couple of prints.

I personally will always try and remove supports before putting the parts into the vats, and definitely before I get to curing, since you are far more likely to scar your print from the support structures if you attempt to remove them after curing.

Remember this process can be messy, but once you scrape the print off and remove supports if you choose, you then place the part into your alcohol bath. I personally have a tooth brush that I use to scrub the outside of the print to make sure I remove the majority of the resin in this first alcohol bath. Make sure you are wearing your nitrile gloves for this entire process. It is smart if the alcohol is moving when you rest the part in it, which is why something like the Elegoo Mercury station can be so useful. That said, I mostly use this first vat to rinse off the bulk of resin and scrub with a tooth brush.

You would then leave the part in the bath for 10-15 minutes, moving the alcohol around periodically, but I do this process in my second alcohol vat. As mentioned, the bulk of the resin will come off and float around in the initial rinsing, which is why I like this 10-15 minutes resting to be in as clean of an alcohol vat as possible. This is especially true when wanting to clean off multiple print plates and you don't want to continually plow through a ton of isopropyl alcohol. Once my first vat gets too full of resin, I will dump it and refill it, and it will then become my second bath with clean alcohol, and my other vat becomes my first.

Make sure your alcohol vat also avoids UV sunlight, since the small resin particulates can cure while in the alcohol vat, which will add to cleanup issues.

Once all the resin is cleaned off after this 15 minutes, you will then want to cure the part via UV light. I have a used cardboard box that I have covered inside with reflective tape, and added UV LED strip lights to the top. I then have a UV powered lazy Susan on the inside. There are quite a few tutorials on making something similar on YouTube, but this entire setup cost me about \$20. All I do is place my part onto the lazy susan, turn on the UV lights, and then close the box. In about 15 minutes the part will be properly cured.

If you don't want to create your own curing station, you can leave your part outside in the sunlight and let the sun do the curing for you. Both of these methods will take about 15 minutes, and you definitely don't want to forget your part and leave it curing for hours. As mentioned in the next section, you can over cure a part and cause it to become too brittle.

Once your part is cured, you will notice it has lost its luster and shine, and transparent parts will become cloudy. If you want this transparency and shine to return, all you have to do is add a clear glossy coat. I have some clear glossy spray paint, and one quick spray makes a huge difference. My cloudy parts all went back to their proper transparency.

Material Notes

As with my overall resin printing experience, I have not tried many resin types. That said, there is a commonality between all resin material types – they cure via UV light. This means I have noticed something that can be quite frustrating- all resins that I have tested have become more brittle over time.

This is because UV light inevitably continues to cure and harden this resin since UV light is near impossible to avoid. I have tested out eSUN's Hard-Tough resin, and boy was that stuff strong right after curing. I was swinging a hammer at prints at full force and they didn't even crack. I could drill into them without any damage to the overall part. And then I tried the same thing two weeks later and all of my prints cracked very easily. This is because the parts inevitably absorbed more UV light, making them more and more brittle.

Because of this, if you want a part to maintain the strength it has right after curing, you will want to hit it with some primer. This is unfortunate, especially if you really like the color and look that the print has right off the build plate, but it is needed to maintain the strength of the part.

There may be methods I am unaware of, but as of writing this book the only way I know to maintain a parts strength is to cover it in some primer as to prevent it from absorbing further UV light over time. This is one reason I also do not suggest using resin printing if your main goal is to have a mechanically strong part.

I have read from individuals who have combined a small amount of flexible resin with tough resin, which has resulted in a very strong part that won't break. I have not personally done tests in this regard so I do not want to comment, but I would assume that the same issue of becoming more and more brittle over time as it absorbs UV light would still be true. Some have said that using roughly 20% Siraya Blu mixed with 15% Tenacious, and the rest whatever color Siraya ABS you prefer, only requires a clear glossy spray and the part will remain strong. Once again, I have not tested this personally, though I thought it worth passing along. Mixing different resins together sounds like it might be a worthwhile experiment in the future, so be sure to follow my YouTube channel if you would like to see that.

Resin printing can be very rewarding and has some advantages over FDM. The photo above shows how it can have much finer detail on smaller prints. But there are just as many disadvantages and more safety concerns. As long as you're safe and have the resources, go ahead and jump into the world of resin 3D printing.



Diagnosing Failures

Don't forget to email me at Sean@3DPrintGeneral.com with proof of purchase for HD photos and PDF.

Nozzle Too Close to Buildplate

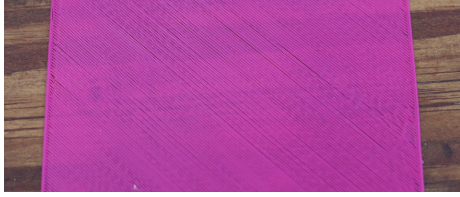


Check Chapters:

[Z-Height Calibration](#)

[Unlevelled Build Plate](#)

Nozzle Too Far from Buildplate

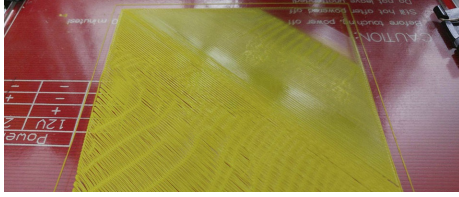


Check Chapters:

[Z-Height Calibration](#)

[Unlevelled Build Plate](#)

Uneven Build Plate



Check Chapters:

[Unlevelled Build Plate](#)

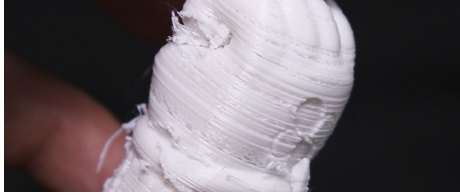
Holes on Sides of Print



Check Chapters:

[Missing Layers and Holes in Print](#)

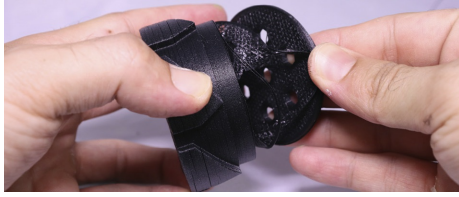
Missing Layers (Temp Under Extrusion)



Check Chapters:

[Missing Layers and Holes in Print](#)

Poor Layer Adhesion



Check Chapters:

[Poor Layer Adhesion](#)

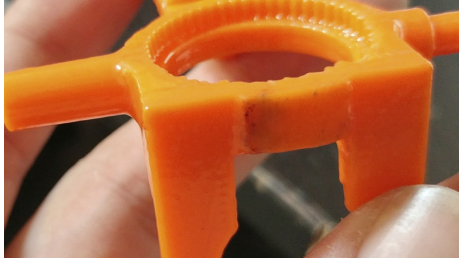
Built Up Material On Nozzle



Check Chapters:

[Built Up Material on Nozzle](#)

Black Spots on Prints

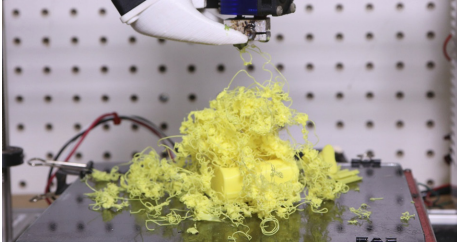


Check Chapters:

[Built Up Material On Nozzle](#)

[Materials and their Settings](#)

Spaghetti Monster



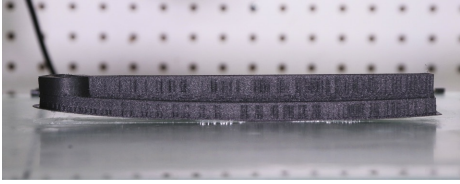
Check Chapters:

[Bed Adhesion](#)

[Parts Being Knocked Over](#)

[Z-Height Calibration](#)

Warping



Check Chapters:

[Warping](#)

[Material Science](#)

[Bed Adhesion](#)

Nozzle Clogs



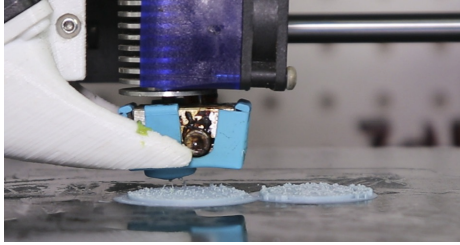
Check Chapters:

[Nozzle Clogs](#)

[Settings Issues](#)

[Bed Adhesion for Above Issue](#)

Ghost Printing



Check Chapters:

[Nozzle Clogs](#)

[Stripped Filament](#)

[Settings Issues](#)

[Materials and their Settings](#)

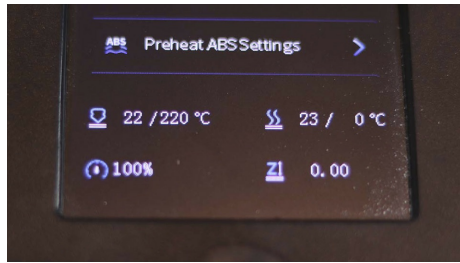
Err MINTEMP or 0° Temp Readout



Check Chapters:

[Hotend Not Reading Correct Temperature](#)

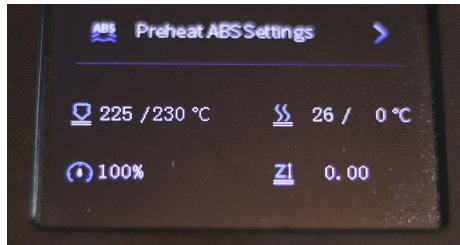
Hotend Not Heating



Check Chapters:

[Hotend Not Heating](#)

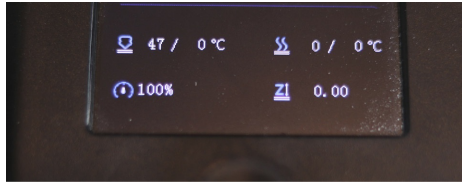
Hotend Cannot Maintain Temp.



Check Chapters:

[Hotend Cannot Reach or Maintain Temperature](#)

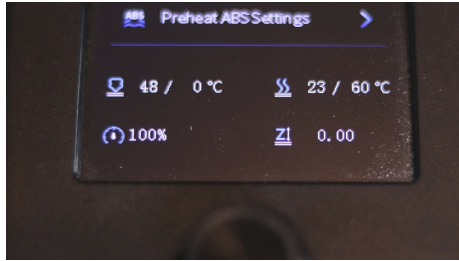
Build Plate Reading 0°



Check Chapters:

[Build Plate Not Reading Correct Temperature](#)

Build Plate Not Heating



Check Chapters:

[Build Plate Not Heating](#)

Layer Delamination



Check Chapters:

[Warping](#)

[Material Science](#)

[Over and Under Extrusion - Under Extrusion](#)

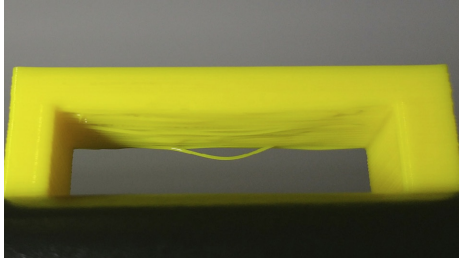
Elephant Foot



Check Chapters:

[Elephant Foot](#)

Droopy Undersides



Check Chapters:

[Settings Issues - Support Settings](#)

[Materials and their Settings](#)

[Material Science](#)

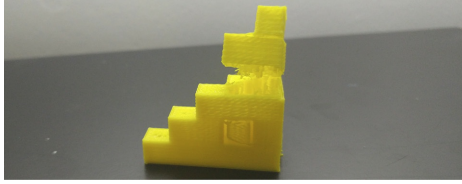
Ugly Underside of Print



Check Chapters:

[Settings Issues - Support Settings](#)

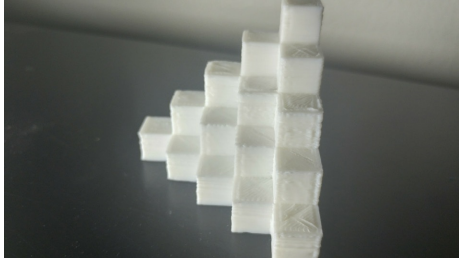
Under Extrusion



Check Chapters:

[Over and Under Extrusion](#)

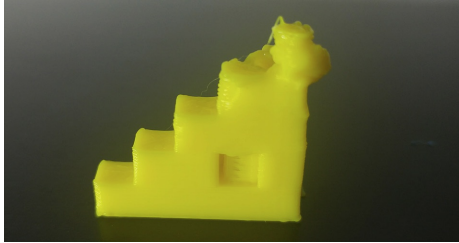
Over Extrusion



Check Chapters:

[Over and Under Extrusion](#)

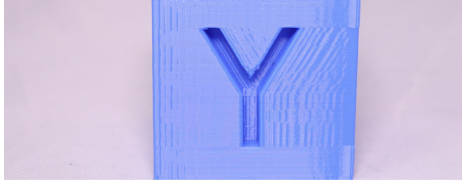
Ugly Top of Print



Check Chapters:

[Settings Issues - Lift Head Cooling](#)

Ghosting



Check Chapters:

[Ghosting](#)

Layer Bulges



Check Chapters:

[Layer Bulges](#)

Text Not Legible



Check Chapters:

[Parts Not Mating Together](#)

[Over and Under Extrusion - Over Extrusion](#)

[Quality Options](#)

Wobbly Print



Check Chapters:

[Z-Axis Wobble](#)

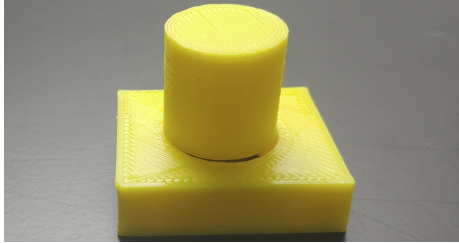
Curling of Layers



Check Chapters:

[Curling of Layers and Angles](#)

Parts Not Mating



Check Chapters:

[Parts Not Mating Together](#)

[Over and Under Extrusion - Over Extrusion](#)

[Quality Options](#)

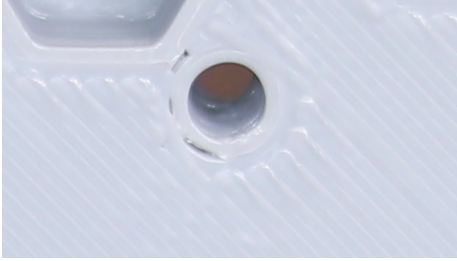
Gaps in Walls



Check Chapters:

[Gaps in Walls](#)

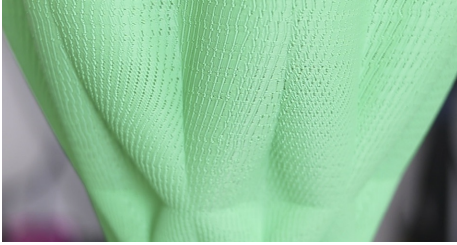
Gaps on Top Layer



Check Chapters:

[Gaps on Top Layers](#)

Vase Mode Issues



Check Chapters:

[Problems with Power Loss Recovery](#)

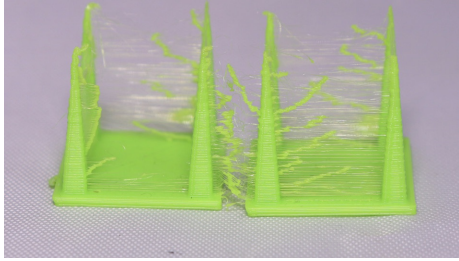
Top Layer Bulges



Check Chapters:

[Settings Issues - Monotonic Top](#)

Hairy Prints

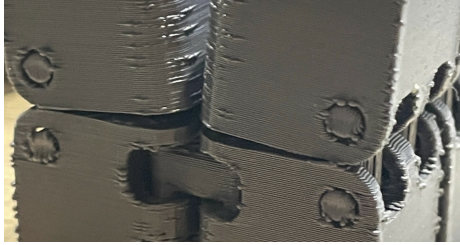


Check Chapters:

[Stringy or Blobby Prints](#)

[Material Science](#)

Zits and Blobs on Prints



Check Chapters:

[Stringy or Blobby Prints](#)

[Material Science](#)

Pitted or Pillowing Top



Check Chapters:

[Settings Issues - Top/Bottom](#)

[Settings Issues - Infill](#)

Single Layer Shift



Check Chapters:

[Layer Shifts](#)

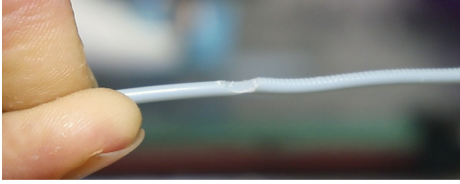
Multiple Layer Shifts



Check Chapters:

[Layer Shifts](#)

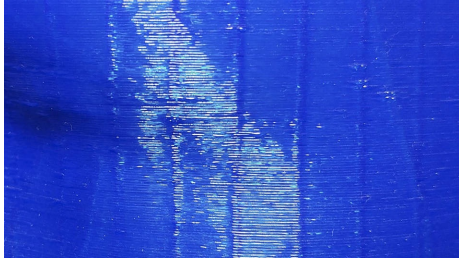
Stripped Filament



Check Chapters:

[Stripped Filament](#)

Veiny Print

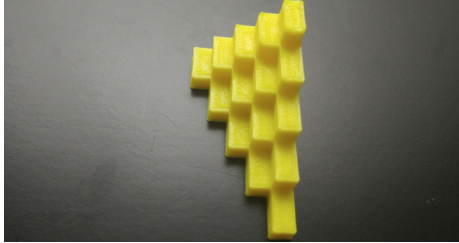


Check Chapters:

[Settings Issues - Infill Overlap](#)

[Ghosting](#)

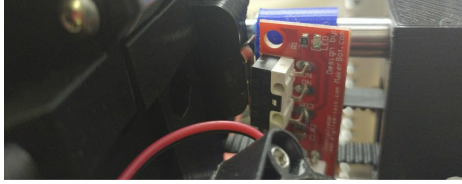
Part Incorrect Dimensions



Check Chapters:

[Parts Not Mating Together](#)

Endstop Not Engaging



Check Chapters:

[Not Finding Home and Inverted Prints](#)

Inverted Print



Check Chapters:

[Not Finding Home and Inverted Prints](#)

Ugly Angles



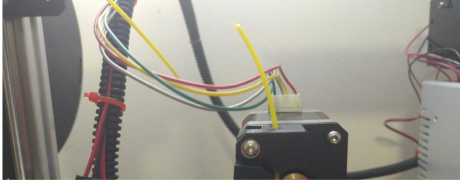
Check Chapters:

[Settings Issues - Support Settings](#)

[Curling of Layers and Angles](#)

[Material Science](#)

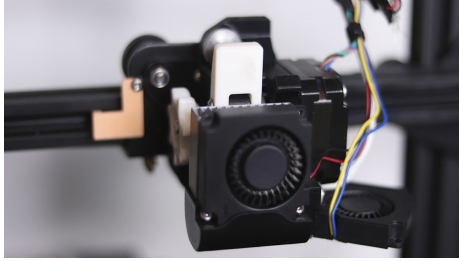
Filament Snaps



Check Chapters:

[Filament Snaps](#)

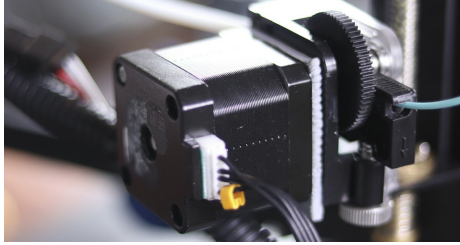
Running Out of Filament



Check Chapters:

[Running Out of Filament](#)

Motor Overheating



Check Chapters:

[Stepper Motors Overheating or Malfunctioning](#)

Bed Adhesion

I am happy that this is the failure that comes first in the alphabetical ordering of my chapters, because having improper bed adhesion is without a doubt the number one cause of failures. While I say bed adhesion, I actually mean every factor that leads to a good first layer, including this chapter as well as the “Unlevelled Build Plate” and “Z-Height Calibration”.

This chapter will be focusing on getting your parts to stick to your particular build plate, though you should definitely reference those two chapters as well for making sure you don't experience a failure 20 hours into a print.

Every single tip in this chapter is crucial for avoiding warping.

What material are you using?

Just about every single material type has different build plate adhesion requirements. That said, even manufacturers can affect this. I personally only stick to reputable brands of materials, since you can run into some really weird issues, including bed adhesion, when the material may have been produced poorly. Please see the “References” chapter at the end of this book for my personal favorites.

I will go over some basic settings for different material types later in this chapter, but if you are working with a material you aren’t very familiar with, I would definitely suggest reading the “Materials and their Settings” as well as the “Material Science” chapter.

Print your first layer slowly with no active cooling fan

This tip is going to be true across all filament types and all 3D printer setups. As you will see throughout this book, I suggest printing slower than most manufacturers advertise. This is particularly true for the first layer.

If there are any issues with that first layer sticking, then it will affect the rest of your print. In order to get it to stick properly, it is best to print very slowly and with no part cooling. Regardless of the speed of the rest of your print, I suggest that your first layer be no higher than 30mm/s. While you may have a setup that can print much faster than this, the first layer is where you should make the exception to ensure that it is properly adhered to your build plate.

Heated build plate

This information is less important now than in previous editions of this book, since you will be hard pressed to find a printer that doesn't already have a heated build plate. This heated build plate is crucial to printing just about any material other than PLA, and even with PLA it helps quite a bit.

For almost all materials the heated build plate should be slightly lower than the material's glass transition temperature. For an understanding of how transition stages occur with polymers used in 3D printing, be sure to read the "Material Science" chapter in this book.

Being near this glass transition temperature allows for the material to adhere better to the build plate, since the material becomes more viscous. Setting the temperature higher than the glass transition temperature can cause "Elephant Foot".

Below are the temperatures I set my build plate at for different material types. Just about every manufacturer should have these stats available to you, though you can use this as reference:

PLA: 55°C

ABS: 100-105°C

ASA: 85-100°C

PETG: 50-80°C

Most TPU: 40-60°C

Nylon: Large range from 40-80°C depending on the formula. Glass with PVA is preferred.

Polycarbonate: 90-110°C with enclosed chamber.

Carbon Fiber Blends: Generally follow whatever the carbon fiber is blended with. 55°C for CF-PLA for example.

Have the proper initial Z-height

Along with this chapter, I recommend reading the “Z-Height Calibration” chapter as well as “Unlevelled Build Plate” chapter. If your nozzle is too far from the build plate, it doesn’t matter what kind of other adhesion methods you have, your part won’t stick and will be knocked off mid-print.

The general starting method for finding your proper z-height is to grab a piece of computer paper, home your printer, and then change the z-height until you get a minor amount of drag on the paper.

Clean your bed before you add anything

This is particularly important when you are using a unique build plate, but having residue on your build plate from a number of different culprits can lead to parts not sticking properly. For PEI build sheets, cleaning is essential if you want parts to continue sticking.

You need to know your build plate before you get to cleaning with anything other than isopropyl alcohol. You can use isopropyl alcohol just fine on PEI. You do not want to use acetone on PEI, since it will destroy the build plate.

You can use just about anything on glass, and acetone will lead to a quick clean. Glass also doesn't need to be cleaned quite as often, and I get prints to stick fine as long as I add a coat of hairspray. I suggest cleaning your PEI build plate every 5-10 prints, and glass every couple dozen. It also doesn't hurt to clean it more frequently than that.

There are quite a lot of build plate options out there now, but it is a pretty safe bet that giving it a good clean with isopropyl alcohol will help. That said, I actually have a build plate from 3DQue that states clear as day that cleaning with anything other than soap and water will destroy the build plate. There is no way for me to know all the possible print bed options out there, so make sure you follow what the manufacturer suggests to clean with if you are afraid of using isopropyl alcohol.

Build plate options

Glass Bed

This is the most basic option available now, but was the preferred method a few years ago. Since I have used a plain glass build plate for so long, I am extremely familiar with it. The benefit to having a glass build plate, especially one that is 1/4 inch or thicker, is that it will not warp when you heat it to 100°C or higher. Having a flat build plate is a huge benefit and I have had magnetic and sticky build plates start to curl up as they heat up to 100°C.

Glass build plates alone will not be enough to get most materials to stick, so here are a few options. My favorite adhesion method for PLA is using Aquanet unscented hairspray. PLA parts stick great, have a shiny underside surface quality, and then pop off with ease when cooled to room temperature. I have used this method for almost 7 years and I still like it, so even if many don't use it anymore, it's worth trying out if you are ever having issues.

This glass bed with hairspray works with PLA, PETG, and smaller parts for most non-nylon prints. When you get to larger parts in these materials, you will likely want a different adhesion method, or perhaps a completely different build plate.

It is smart to clean your hair sprayed glass build plate periodically. To be honest, I sometimes let this go for dozens of prints before cleaning, so cleaning isn't as necessary as it is with PEI. That said, it doesn't hurt to clean everything off with some isopropyl alcohol and a scraper. Then reapply the Aquanet hairspray to the clean glass.

The main method years ago was to just add blue painters tape to your glass build plate. To be honest, this is more needed for printers that do not have a heated build plate. That said, if you are ever having difficulty getting your part to stick and you've tried it all, the tried and true method of blue painters tape should just about always work. Just remember after adding the blue painters tape, you will need to raise your z-height slightly as to not smash the nozzle into the tape.

When it comes to printing nylon on glass, I recommend a PVA mixture. I am not familiar with all build plate types, but I do know that PEI build plates don't work great with most nylons. Most nylons are very hard to print without warping, and so what I will normally do is grab some Elmers glue, mix it with water in a 1:1 ratio, and then paint it onto the glass build plate at room temperature. I then heat the build plate to whatever the nylon calls for (most are between 45-60°C) and wait until all of the water is evaporated. This

can take around 10-15 minutes depending on how hot your build plate is.

This PVA mixture will work on multiple material types, not just nylon. The same is true for other PVA based items like glue sticks. The underside of your print won't be quite as pretty if you use a glue stick, but it is something that I have definitely used from time to time, especially when using a TPU material that isn't sticking super well to the build plate.

When it comes to working with ABS and ASA filaments on a glass build plate, I used to recommend making an acetone slurry. I have recently discovered a product called Magigoo that works far better than this slurry. Magigoo has been around for a little while and it works wonders, especially with ABS and ASA. Parts stick great when the build plate is heated, and they come off easily when at room temperature.

Magigoo Original works for PLA, ABS, ASA, PETG, Hips and TPU. Their original formula is only \$16 on Amazon and will last you a very long time. You just spread some on your build plate, and watch warping disappear. You will be shocked at how it just slides off when the build plate cools down. Cleaning is just as easy with a few wet paper towels and some scrubbing, and it comes off far easier than something like an ABS slurry.

Magigoo also has formulas specific to Nylon, Polycarbonate, Polypropylene, TPU, and PEEK. Other than their original formula, I have only personally tested their Polycarbonate version, and it works great. Not quite as great as their standard formula does with ABS, but far better than any other method I have tried with Polycarbonate. Keep in mind you will still want an enclosure with high ambient air for large ABS, ASA, and Polycarbonate prints, but your build plate adhesion will not be your limiting factor.

While I haven't tried all their blends, I am likely going to have to try their nylon product since nylon is one of the hardest materials to get to stick properly. If it works half as well as their other products, it will be worth it.

You can also use Magigoo on PEI build plates as well if your parts just aren't sticking.

PEI Build Plate

There are so many versions of PEI build sheets now that it's difficult for me to only speak of one generic version. In the past I wouldn't have recommended a PEI sheet, since they would get damaged easily. These days, companies like BuildTak, Wham Bam, Fulament, and TH3D all have PEI coated flexible build plates that have nothing but great reviews. I personally used Fulament's version in a review and it definitely works well. I don't have it on an enclosed printer, so it is hard for me to test large ABS parts.

Everything I have tried thus far though sticks great and pops off easily with a quick flex. Also, there seems to be no fear of breaking the PEI or damaging it.

There is even a smooth sided PEI side as well as a textured one. So it is very easy to add some Magigoo to that smooth side for ultimate hold.

The other option is a full PEI build plate, not just a sticky sheet. These are a bit more expensive and are not quite as common as I remember them being, but they worked great when I used them.

PEI creates very small suction cups as the build plate heats up, which allows for a good bed adhesion. Then as the bed cools to room temperature, the suction cups shrink, releasing its hold on the part. This means parts stick great without the need for any additional bed adhesion. In fact, you can damage the PEI if you add adhesives, so you do not want to use anything on it.

The one thing that is needed when printing on PEI build plates is to clean them frequently. PEI loses adhesion quickly if it isn't cleaned; so quickly that within 10 prints you might be battling with nothing sticking. So I suggest around every 5 prints to clean off the build plate with some isopropyl alcohol, not acetone.

You want to make sure you have the proper z-height honed in before starting a print, because if your nozzle is too close you can damage your nice PEI build plate or sheet. The new Fulament flex plate I have seems very tough to damage but it is still smart to have the proper height honed in first.

The flex plates by the four companies I just mentioned have become the preferred method within the community. So while I may use glass with hairspray, I stress that you research one of these options if you plan on printing high temp materials frequently.

Other Build Plates

When I started 3D printing, there were only glass and acrylic build plate options. Then PEI eventually came out. Since acrylic is terrible to print onto, I got used to using plain glass. These days though just about every printer comes with its own build plate style. The Ender 3 V2 (and similar clones) comes with what is called a carborundum glass bed.

Prints seem to stick well to the texture, though I have used hairspray a few times. The glass chipped after a few dozen prints so I no longer use it on my Ender 3, but overall this should be better than standard glass.

The Ender 5 comes with a very thin magnetic sheet, and works a bit better than I thought it would. FlashForge printers have a sticky sheet that reminds

me of blue painters tape. Qidi has magnetic plates that parts stick phenomenally to, sometimes too well. There are also endless options that I have not tried.

It is worth trying whatever comes with your printer before you go out and purchase a flex plate, because if you are happy with it, there is no reason to spend extra money.

Using a brim

A brim refers to lines that follow the perimeter of your print that essentially act as an anchor for your part. I do not use a brim at all for PLA or other non-warping materials though, since they can be a nuisance to remove. But for high warping parts I do recommend using one. For materials like ABS and ASA, the brim is extremely easy to remove.

Please refer to the “Settings Issues” chapter for a full explanation, but you can turn your brim on within your slicer. Every slicer is a bit different but it is generally in a “build adhesion” section where you can choose between a brim, skirt, or raft.

How thick the brim is will be based off of your nozzle diameter. A brim of 15 lines will be twice as wide with a 0.8mm nozzle as it would be with a 0.4mm nozzle. For most parts and nozzle diameters requiring a brim I will use anywhere from 10-30 lines. Anything more than 30 is likely unnecessary.

I am not a big fan of using a raft unless I have a lot of small parts to print, but that is another option as well, as explored in the “Settings Issues” chapter.

Initial layer thickness (horizontal expansion)

Initial layer thickness, or horizontal expansion depending on what slicer you are using, will cause your first layer to be thicker than the rest. While this helps with build plate adhesion, it can add to issues of “elephant foot”. If you really need a part to stick well and you don’t care much about the bottom layer being slightly thicker than the rest of your print, you can increase the horizontal expansion to increase the bed adhesion.

You can also set the horizontal expansion to a negative number as to reduce the chances of elephant foot.

Initial layer height

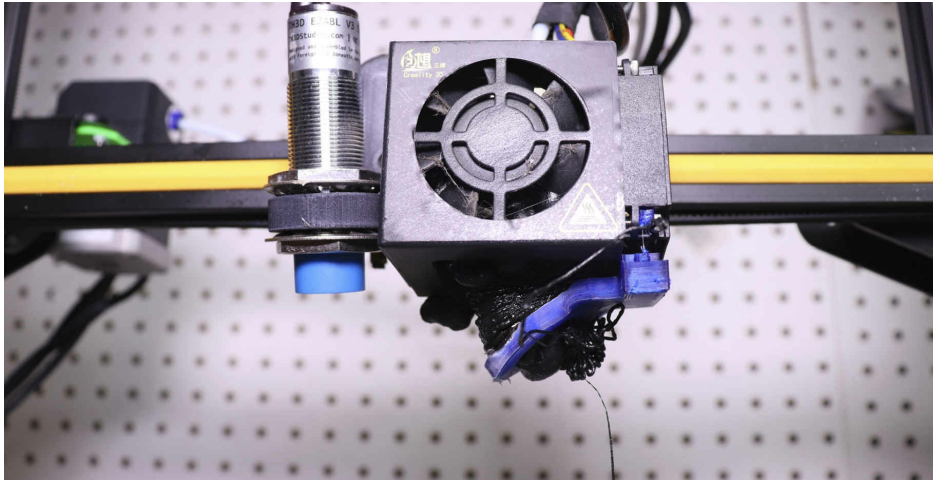
Rather than tweaking the thickness of the initial layer, you can tweak the initial layer height to help the first layer print properly. This is a very important thing to utilize, especially if you are printing the rest of your part at low layer heights.

It is much harder to dial in the proper z-height and to level your build plate with a 0.1mm first layer versus a 0.3mm first layer. Having a larger first layer allows for a lot more room for error in either a deformed build plate or just a slightly off z-height.

Whenever I use a smaller diameter nozzle, like 0.25mm or even 0.15mm, I am limited to how thick my initial layer can be. Because of this, one of the most annoying things about using a smaller diameter nozzle is dialing in the perfect z-height.

Save yourself a headache and increase your initial layer height to be right around the max of your nozzle diameter (roughly 75%). So if you are using a 0.4mm nozzle I suggest your initial layer height be 0.3mm, regardless of what you have set for the rest of your print.

What to do if this happens



This, along with the “spaghetti monster”, can happen if you start a long print and it gets knocked off the build plate. The print will continue and could result in this ugly, frustrating mess.

The way to prevent this from happening is by following all of the precautions above. I also cover some methods to cleaning this mess up in the “Parts Being Knocked Over” chapter.

Summary of Fixes and Precautions

- Know what material you are using as well as what is required for it.
- Heat your build plate to either near the glass transition temperature of the material you are using, or to a specific temperature suggested by the manufacturer.
- Frequently clean your build plate, especially if you are having bed adhesion issues. Use Isopropyl alcohol for most build plate types, though you can use acetone on glass. I suggest only using water and soap for cleaning off PVA or if your build plate is sensitive like the one from 3DQue.
- If you are using glass, you will want to add extra adhesion, such as Aquanet unscented hairspray for PLA, PETG, and small ABS parts.
- Check out Magigoo; I highly recommend their products.
- I suggest checking out the build plate options by Wham Bam, BuildTak, TH3D, and other competitors as an alternative to regular glass. There are quite a lot of options, and many printers come stock with an upgraded build plate now.
- Hone in the initial z-height by referring to the “Z-Height Calibration” chapter.
- Slow the speed down and turn off the active cooling fan for your first layer on every material type.
- Use a brim to help anchor the part if printing a higher warping material.
- Print with a raft if a brim isn’t enough, though I normally only use rafts for lots of small parts on a single build plate.
- You can increase the initial layer thickness, though it will distort the dimensions of the bottom of your print.
- Increase the initial layer height to max out your nozzle diameter (75% of the nozzle diameter), so that the tolerances of your initial Z-height is a lot easier to hone in.

Build Plate Not Heating

This issue is very easy to diagnose since your build plate will not heat up when you tell it to. This should not be a thermistor issue, but rather one of two issues: your heater cords or your board overheating.

Confirm you are using the correct volt/amp for your heater/board before moving forward. If you purchased your printer from a manufacturer and didn't build it, then you shouldn't have to worry about that.

Burnt out or disconnected heater wire

Over time, especially with Cartesian machines where the build plate rattles back and forth, these wires can start to get worn out or potentially tugged from their connectors. Along with constant movement, your build plate is receiving the majority of the power your printer is providing, since heating up large surfaces takes a decent amount of electricity.

I would first visually inspect the thick wires connected to the underside of your build plate that is wired to your board to see if anything has become disconnected. Then look for any black or burnt areas. This is the easiest way to find the culprit. If you see an issue, with your printer off and power cord disconnected, reconnect wires or cut off the burnt section and then re-solder the two wires back together. Keep in mind you just made the wire shorter, so confirm it can reach all points without restricting your build plate. If it does, you unfortunately will need a new wire, or need to solder on an extension.

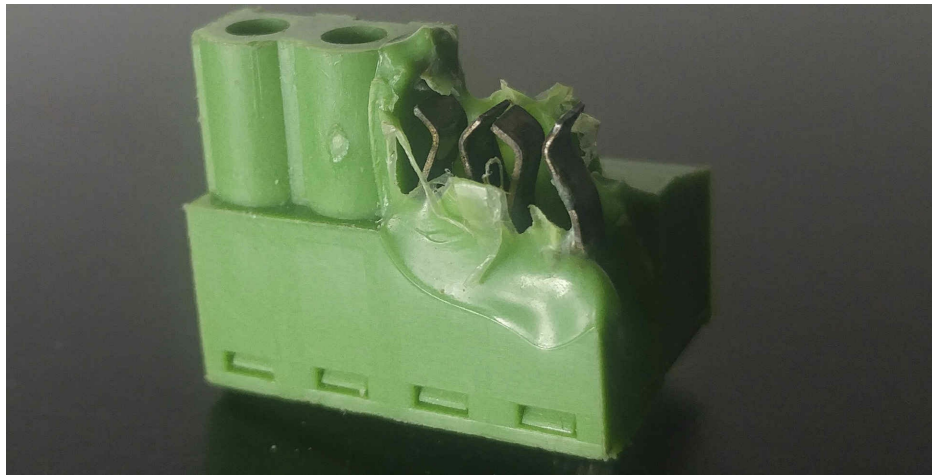
If you do not visually see any issues, pull out your multimeter and check for connectivity from the build plate to the board. Please see the “Important Accessories and Replacements” chapter for a list of items I suggest to use. If you do not have connectivity in one or both wires, you will need to replace them if you are not easily able to figure out which section of the wire has gone out.

DO NOT replace a heater cord with a normal, small gauged wire (thinner than 14 gauge). You will be susceptible to a fire and further burnt out wires.

Burnt out connectors on board

This used to be very common on inexpensive RAMPS boards, though they are far less common now. I do not suggest using a RAMPS board unless you want to play around with a very inexpensive build. If you are using a RAMPS board, these small green connectors are not rated high enough for the amount of current.

If you are using a RAMPS board, you need to upgrade this connector to a well-made 16A replacement (which can be found at DigiKey or Radoshack). This should not be an issue on any pre-built machines sold today.



Regardless of if you are using a RAMPS board, I would still check the connector that your build plate heater is connected to in order to see if there is any disconnection or anything burnt out.

Board overheating

This problem normally shows itself in a different manner than the previous explained issues, because you will likely see the build plate start to heat, maybe even reach its target temperature, and then randomly turn off. If it takes a couple hours to show itself, you may not even notice it happened until your part gets knocked off or warps badly with a room temperature build plate.

Any pre-built machine should come standard with at least one fan blowing on your board. First thing you should do is make sure this fan is working. If you built your own machine, you need to add at least one fan blowing onto the board.

If the fan is not working, check to see if it is the wires that are not working, otherwise you will need to replace it.

Once again, if you are using a RAMPS board, you will likely want to upgrade it. At a minimum, if your RAMPS board has thousands of hours of printing, you should at least buy a \$10 replacement, then make sure it is being properly cooled.

If you are purchasing a printer from a reputable manufacturer you should not experience these issues, though you do want to confirm that the board fans are blowing properly regardless.

Purchase a new heated build plate

This should not be needed in over 99% of cases where your build plate stopped heating, since a heated build plate is just a simple resistive element. I honestly cannot remember the last time I had to replace the actual heater on the underside of my build plate.

If you have confirmed that your wires have connectivity from the build plate to the board, that your board connectors look fine and everything is secure, and your board is being properly cooled, then it may be the rare case that your actual heated build plate is malfunctioning. Double-check the sections above before deciding this, since this can be an expensive purchase.

If your printer is new and from a reputable manufacturer, email them to see if they can offer a replacement.

Build plate will only heat

I decided to throw this in even though it is a bit of the exact opposite problem as the rest of this chapter, I just wasn't sure where to include it. If your build plate decides to start heating when you turn on the machine, without telling it to heat, then you have a faulty MOSFET. The MOSFET that controls the board failed, and when they fail they normally turn to "ON", which means the board will attempt to heat right when you turn on the printer.

If the MOSFET is attached to your board, you will unfortunately need to swap the entire board. If the MOSFET is external, you can go ahead and change it.

Summary of Fixes and Precautions

- Confirm you are using the correct Volt/Amp for your heated build plate/board.
- Check if a heater wire got disconnected from the board or build plate. Reconnect if found.
- Visually inspect the heater wires for any burnt out sections. If found, cut off and re-solder the two wires back together.
- If you cannot visually see any issues, pull out the multimeter and check connectivity of both heater wires from build plate to board. Replace any wire not showing connectivity.
- Inspect the board for any burnt out connectors or wires. Replace if needed, and upgrade if using a RAMPS board.
- Actively cool your board and make sure any fans blowing on the board are working properly.
- Replace board if overused or burnt out.

Build Plate Not Reading Correct Temperature

At times this problem can be difficult to diagnose and if left unattended it can lead to some serious issues.

These instructions are very similar to the “Hotend Not Reading Correct Temperature” chapter.

Build plate reading 0° or you receive a “bed not heating” error

There is a thermistor for your heated build plate that works as a thermometer - and just as with the hotend's thermistor, it can become damaged or disconnected. These thermistors are not very expensive, but can be difficult to replace on certain setups. So while it is good to have a spare, you will likely want to test everything else first.

If your thermistor has no physical damage that you can see, you will want to check for continuity in the wire. If there is a frayed wire, or a section of a wire you diagnosed as having no continuity via a multimeter, you can either cut out the damaged section and solder, or rewire entirely. If your thermistor is still intact, replacing the wiring will likely fix your issue. When repairing the wiring you will then need to confirm that the build plate can reach its farthest point from the board, since you just made the wire shorter than it was.

If your newly soldered wire cannot reach the board at the build plate's furthest point, you will experience layer shifts on large builds, or just perhaps a repeat of the problem as the wire rips out again.

Note: If you are in a pinch, soldering the thermistor line will work but it may lead to additional process issues since you are changing the resistive value of that line and therefore the temperature that the board is reading. Your material profiles may require slightly different temperatures after doing this. It is typically recommended to completely replace this section of the wiring if possible.

Confirm thermistor is attached properly

Well-built machines will have their thermistors attached snugly to the build plate, but cheaper machines may only be held onto your heated platform by some Kapton tape. This isn't as common on newer machines, but I have definitely used printers in the past with thermistors just held onto the build plate with some tape. Constant moving and shaking of your build plate can make it so that this tape becomes disconnected, and your thermistor will inevitably shake out of its holder.

This will cause your thermistor to read a lower temperature than your build plate actually is, since it is not actually touching it. This problem can become severe if the thermistor gets far enough from the build plate, which can cause your heated platform to continue to rise in temperature until your board overheats.

You need to confirm that your thermistor is attached properly to the build plate and that there is no chance of it being ripped out mid print.

Having a build plate that is off by 5 degrees typically will not affect the quality of your print, but it can definitely become an issue when the differential becomes higher than this.

Still experiencing issues

If you are still experiencing issues, or you notice that your bed is continually 10 or more degrees off from where it should be, you will likely want to replace your thermistor and rewire it from the build plate to the board.

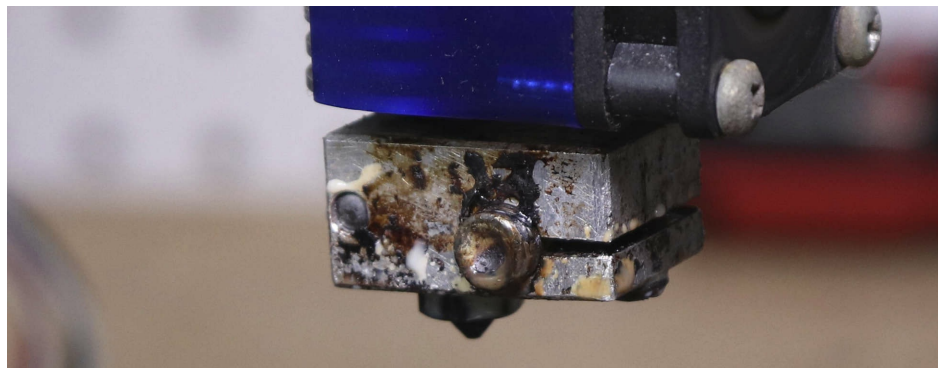
If the problem continues, flash your machine with the original firmware, and then replace the board if necessary. Replacing the board should be a last-case solution.

Summary of Fixes and Precautions

- Make sure the thermistor is actually connected to the build plate and not hovering next to it.
- If thermistor is noticeably damaged, replace it.
- Check for breaks or frays in your wire and solder or rewire as needed.
- Make sure that if you do rewire, you give enough slack to allow the build plate to move to its farthest point.
- Replace and rewire thermistor from the heated platform all the way to the board.
- Flash firmware to factory settings.
- If it's still malfunctioning, replace board.

Built Up Material on Nozzle

If you do not have a specific hotend for every material you are using, you will likely experience some black dots on your prints from time to time. Even when you are using only one material, this can still be a frequent occurrence. This can be from a few culprits, but often it will show itself by having built up material on your hotend.



This issue hasn't had any real innovations since the last edition of my book, so this chapter is pretty similar in its diagnosis.

One of the biggest issues with this failure is that you will often not be able to diagnose it until it happens. This means that a black spot might show up on an important section of your print 10+ hours in. This is why it is crucial you maintain the cleanliness of your hotend by frequently purging and cold pulling any residue from your nozzle, along with using a nylon brush and silicone sock.

Ensure your hotend and nozzle is set up properly

Every hotend setup needs to be assembled in a slightly different fashion, but nearly all of them require you to not over-tighten.

When the heater rises in temperature, the metal expands and can cause your once tight nozzle/heater block to have minor gaps. This gap can cause material to ooze out and make its way onto your print. Since this material has been stuck on a hot nozzle before finally being pushed onto your model, it will likely be black and burnt, regardless of the color you are using.

If you notice that your heater block is loose when hot, or that you constantly have to brush off the nozzle or hotend from excess material, you will likely need to tighten these parts.

I always suggest doing the final tightening of your nozzle and heater block when heated to 240°C or higher (if using an all-metal hotend), and using proper gloves and tools. Remember that you have a high chance of burning yourself, so do this with caution.

You also want to make sure to not over-tighten anything. I have broken quite a few heater blocks, nozzles, and heat break barrels due to over-tightening. These parts, especially when hot, can easily snap under pressure. When you are doing this, make sure to only tighten until you know that the nozzle and heat block are not loose, and will not unscrew during printing.

If you are still experiencing material oozing from the gap between your nozzle and heater block you will likely need to upgrade or replace your nozzle, heater block, or entire hotend. Poorly made or worn out parts will not have tight tolerances, and can lead to these gaps in your threads. I have seen images of cheap knockoff products cut in half showing just how poor their tolerances are.

This is why it is important to only purchase name brand parts and not to buy aftermarket knock-offs. For example, if you want an E3D hotend, only purchase from verified dealers such as E3D, Filastruder or Matterhackers, because there are many counterfeit products on the market. Matterhackers is a good source for most 3D printing parts.

Purging material

Every time you switch filaments or after very long prints, you should purge out the material that may have oxidized inside of the hotend. There are a couple of ways that you can do this.

If you are using the same material, you can go ahead and heat the hotend to its printing temperature. Then push down the filament for about an inch, and pull up quickly. Cut off the end and you should be good to go with your next print. Repeat this step as necessary if you are switching to a different color in order to ensure that you do not get any mixed colors during your print.

If you are switching to a material that prints at a higher temperature (such as switching from PLA to ABS), purging is normally simple. Do the same procedure as above, but multiple times in order to confirm that there is no remaining residue within the hotend. If this hotter material is in a different color, then you will follow the same procedure, it will just be more apparent when you haven't purged enough. Since you are printing at a higher temperature, the majority of the previous material should be removed.

A real issue occurs when you are switching to a material that prints at a lower temperature than your previous filament (such as switching from ABS to PLA). It is likely you will not be able to purge all of the residue material with the above method, because the ABS needs to be purged at a temperature it can properly extrude at.

If you like to live dangerously, you can purge this material by pushing the colder filament through the hotend when it is set to the higher temperature (such as extruding PLA at 245°C when switching from ABS). Push the material through at a steady pace and then pull it out very quickly, making sure to not let it sit. If you attempt this method, you are going to have a higher chance of a nozzle clog, and you may not get 100% of the residue material.

The proper way to get rid of this material would be to do a Cold Pull as described below, or to purge by using a cleaning filament/nylon material.

Purging with cleaning filament or nylon can be done by heating your extruder to a temperature of around 240°C – 250°C (or whatever the material you are using calls for), and then extruding the cleaning filament through as you would with the examples above. Quickly pull the filament out as to not leave any residue, and you will see just how much gunk the cleaning filament was able to pull out.

Though less common when using cleaning filament than with other printing materials, you still run the risk of leaving residual material in the hotend that

will come out later as a black, burnt spot. That is why the best method is actually a cold pull.

Cold pull

When switching to a material that prints at a lower temperature than your previous filament you will likely want to do a cold pull. Cold pulls are also very beneficial to do as regular maintenance on your machine regardless.

I personally like to perform cold pulls with either a cleaning filament or Nylon mix, but you can perform them with the material you are trying to clear out. My favorite material to do this with is Nylon 910 by taulman3D. Not only is that material great to print with, it seems to work even better than cleaning filament I have used in the past for removing the oxidized material in the hotend.

What you do is heat the hotend to the temperature of the material you are using to do the cold pull (250°C for Nylon 910). Push the filament through for an inch, or as much is required for you to no longer see the previous material coming out the nozzle.

Then quickly set your hotend to 130°C - 150°C (I normally do 130°C). You don't want to leave the material sitting in the hotend for a long period of time because it can oxidize itself, or even cause heat creep in your barrel. Once the nozzle cools to this newly set temperature you will want to pull out the filament. This can be difficult if there is a lot of built up residue material, but it normally doesn't require too much effort.

Once you pull you should see excess burnt or colored material on the filament you just cold pulled. Repeat this process until you no longer see this residue.

This is the best way I know of, other than purchasing a new hotend/heater block to get rid of the excess and oxidized material.

Excess oozing

I recommend reading the “Material Science” chapter, particularly where oozing is discussed, to help prevent excess material from coming out of the nozzle.

Use a wire or nylon brush to clean nozzle and hotend

You should have a brass or copper brush on hand to clean the nozzle and hotend periodically. Before a print, especially one where you see material has built up on the heater block/nozzle, you will want to clean it with a wire brush (when heated). When the nozzle is hot you can brush off any excess material that has built up.

Many people suggest using a nylon brush instead of a wire brush since it is far less abrasive. I frequently use a nylon brush and only periodically use a wire brush on a very dirty hotend. If using a metal wire brush it is better to use one made of copper or brass as opposed to steel to reduce the amount of nozzle abrasion that occurs. Keep in mind that a normal nylon brush will melt on the heated hotend if left on for any longer than a quick wipe.

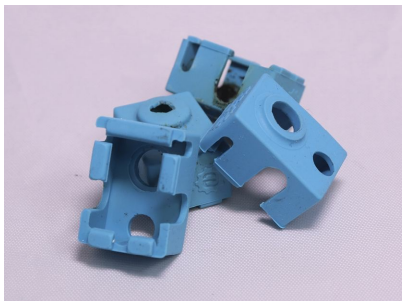
This is crucial in maintaining a clean nozzle and reducing the amount of burnt spots experienced on a print.

Don't leave filament resting in a heated hotend

You shouldn't heat your hotend until you are ready to extrude. If you leave filament in a heated hotend for long periods of time you will increase your chance of nozzle clogs and oxidization of the material.

Make sure all of your end G-codes have the script M104 S0, which turns off your hotend after completing a print.

Use a silicone sock if available



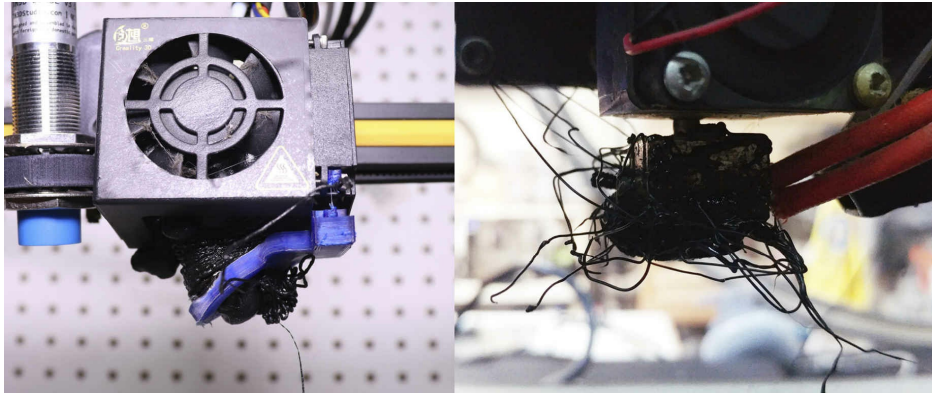
One great improvement done by E3D for their hotends was adding a simple silicone sock for the heater block and nozzle. This was such a great improvement that just about every single hotend on the

market now comes with a sock. It is rare to find a hotend without one, but you will definitely want one on hand if yours is worn out or you do not have one.

As mentioned in the “Hotend Can’t Reach or Maintain a Temperature” chapter, these socks help to keep the hotend from sporadically changing temperatures. The best part about these socks though is the extreme reduction in black, burnt spots on your prints. Your heater block and nozzle will remain in shiny clean condition so long as you print using the sock.

Black, burnt spots are still possible due to the fact you can have oxidized material built up in the heater block, but the problem should be vastly reduced.

Excess material built up on nozzle



The two examples shown above have an extreme amount of material stuck to the nozzle. This is caused by letting a part that is not stuck to the build plate continue to print. Once a part is knocked off or does not have the proper bed adhesion, your printer will continue to print and material will build up on the hotend.

Please refer to the “Parts Being Knocked Over” chapter for ways to clean this and “Bed Adhesion” to prevent it, since this isn’t exactly the same problem described in this chapter.

Summary of Fixes and Precautions

- Make sure you have a well-made hotend and that everything is tightened when heated to 240°C or higher. Take proper precaution to not over-tighten.
- Purge old material by pushing down new material an inch and pulling out quickly. Cut off any old material that is stuck onto the filament and repeat the process until there is no longer any excess residue.
- Use cleaning filament or Nylon 910 since it works best for this process. If not, you can use the material you are printing with.
- Cold pull by extruding cleaning filament/Nylon through the hotend at 250°C. Allow the nozzle to then cool to 140°C – 160°C, and pull the filament out. Repeat this process until you no longer get discoloring.
- Don't leave filament in a heated hotend for extended periods of time.
- Use a silicone sock to reduce black spots on your print.

Curling of Layers and Angles

This problem can mimic what looks like warping, but rather than the part warping off of the build plate, the individual layers seem to be warping upward, which I refer to as curling. You will see that the solutions to this problem are nearly the exact opposite of the ones given in the “Warping” chapter, since this curling issue doesn’t actually have anything to do with warping.

Turn on Active Cooling Fan

This problem of curling is most common when you print in PLA without an active cooling fan, or with your active cooling fan not turn high enough. Whenever I print in any PLA option, I will always have my active cooling fan turned on to 100% after the first layer is printed.

The photo below shows a PLA print with the same settings, but the one on the left did not utilize an active cooling fan, and the one on the right did.



While you may hear me say in other places in this book that an active cooling fan may reduce layer adhesion on some materials, this is not true for PLA. Printing PLA without an active cooling fan will inevitably have an ugly look to it, and have curling of top layers and angles.

This is also true for other particular materials, so you need to reference your manufacturer's guide for whether an active cooling fan should be used or not for the filament you are using.

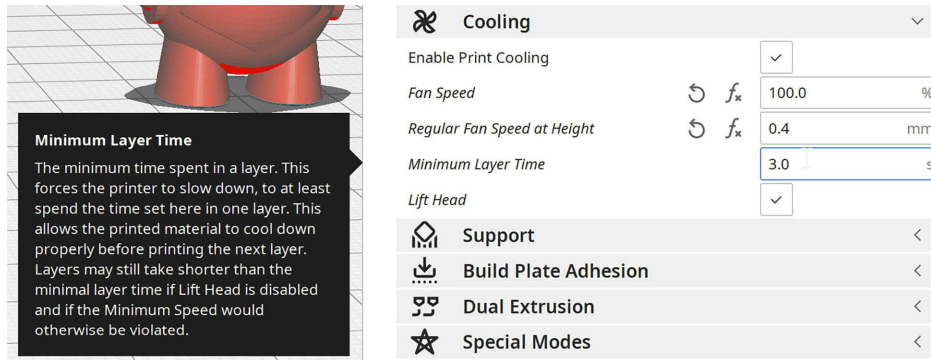
Too hot of an environment

This problem of curling could also arise if the actual environment is too hot for the material you are using. You would not want to print PLA in an enclosed chamber, since the ambient air will get too hot for the material to cool properly, even if you have an active cooling fan on. PLA prefers to be printed in an open environment, especially due to its low glass transition temperature.

The same is true for some other materials. In general, if the material you are printing does not have difficulty with warping, then you will not want to print it in an enclosed chamber. If your printer is enclosed, you will want to either remove the lid or leave the printer door open when printing in PLA.

Not enough time for layers to cool

If you are printing a small section on your print in which each layer prints on top of each other in quick succession, then the previous layer may not have been given enough time to cool. This is why the “Minimum Layer Time” setting exists in Cura under the “Cooling” section.



In general, the majority of materials will be fine with a 3-5 second minimum layer time. So if I were to set this to 3 seconds, and a layer completes in under 3 seconds, your printer will pause and wait before starting the next layer. If your layer were to complete in 1 second and then start the next layer without pausing, then the previous layer will still be hot and start to curl upward as the nozzle goes back over it.

It is advisable to also check the “Lift Head” box in Cura along with setting the “Minimum Layer Time”.

Too slow for layer height

If you are printing at a very slow speed, particularly for low layer heights, then it is possible your layers will start to curl upward. Lower layer heights have less rigidity than larger ones, meaning this problem can be more likely. A 0.3mm layer will be much more rigid than one that is 0.1mm, and be less likely to curl upward.

You can help mitigate this curling issue by either increasing your layer height or increasing your printing speed.

Too hot of an extrusion temperature

You can also have curling of your layers if you set your printing temperature too high. Make sure you are printing within the range of your particular material. This is particularly true when printing with smaller nozzles and layer heights, since less volume of material is extruding per second when compared to larger nozzles and layer heights.

When printing with low layer heights on small nozzles, reduce your printing temperature to the lower range of the manufacturer's suggestion if you are experiencing curling of layers.

Turn on support structures

This curling can happen on unsupported angles, especially on materials that cannot use an active cooling fan. When I print in ABS or ASA, I cannot use an active cooling fan as to reduce the risk of warping and delamination. This means that angles cannot be printed nearly as well as PLA with its use of an active cooling fan.

When printing in PLA, I generally have my support structures set to print at 55 degrees (with minor tweaks for different layer heights). When I print in ABS, I set my support structures to 45 degrees. This means ABS will require more support structures than PLA for particular angles. If I were to set my ABS prints to only require support structures at 55 degrees, than all angles between 45 and 55 will likely have curling.

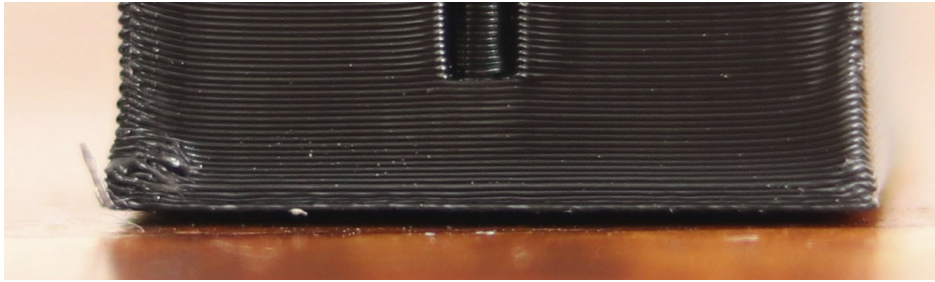
Moisture in material

As with other problems that could arise, moisture in your material can also lead to this problem of curling. If you have tried all of the previous solutions, and your layers are still curling upward, then you will want to dry your spool. I have instructions of how to do this under the “Materials and their Settings” chapter, as well in the “Stripped Filament” chapter.

Summary of Fixes and Precautions

- Make sure your active cooling fan is on if your material calls for it. 100% fan speed is recommended after the first layer when printing in PLA.
- Make sure the ambient air is not approaching the glass transition temperature of the material you are printing. PLA should not be printed in a fully enclosed machine.
- Have your minimum layer time set to at least 3 seconds in Cura so that each layer has enough time to cool.
- Low layer heights have less rigidity, meaning increasing your layer height or your print speed will help.
- Reduce your printing temperature, especially if printing at low layer heights on small nozzle diameters.
- Turn on support structures and reduce the angle for when they are required, particularly with materials that cannot use an active cooling fan.
- Remove moisture in your material if you tried all of the above methods.

Elephant Foot



Elephant foot is an issue where the bottom few layers of your print are much thicker than the rest of your print.

This is a fairly straight forward issue to fix as there can only be a couple of causes.

Nozzle too close to build plate

This issue is also covered in the Z-Height Calibration chapter, but can result in an ugly elephant foot if not dealt with. When the first layer has the nozzle too close to the build plate, material is built up, smashed out, and presents itself as thicker than the dimensions of the actual part. Without enough distance between the nozzle and the build plate, this issue is going to be hard to avoid.

The elephant foot will correct after about 5-10 layers, but the bottom section of your part will definitely be the incorrect dimensions. Refer to the Z-Height Calibration chapter to fix this.

Build plate too hot

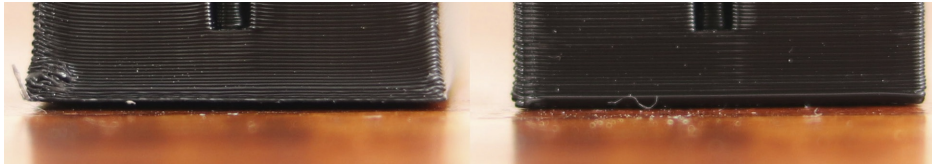
Another reason an elephant foot failure can occur is from running your build plate too hot for the material being extruded. I never run PLA with a build plate hotter than 60 degrees Celsius (sometimes only 50 degrees), but if you do, you can have a distorted bottom of your print.

This is because you are setting the build plate higher than the materials' glass transition temperature. This means that the material on the bottom few layers becomes deformed as material is deposited on top of them. While it is easier to get good bed adhesion at these high bed temperatures, the deformation causes this elephant foot.

Make sure you are using the proper build plate temperature for the material you are using by referring to the manufacturer suggestions. If you know you have the proper Z-height and are operating within the suggested temperature range, and are still getting an elephant foot, you should attempt reducing your build plate temperature a bit further. Otherwise you can try out the next suggestion to make sure this problem is eliminated.

Use a raft

It is rare that I use a raft on my standard DIY machines, but if elephant foot is a consistent issue, a raft should make this failure disappear. A raft can fix having your nozzle too close to your build plate, and it can also fix having the build plate be too hot. The raft acts as a barrier between your print and the bed and should mean you no longer have any elephant foot issues. The photo below shows a print with elephant foot on the left, and a print that used a raft on the right.



Many printers are meant to use a raft standard in order to help with bed adhesion, and so long as the settings are dialed in, a raft can be a great solution.

To read more about rafts and the proper settings for your slicer, refer to the “Settings Issues” chapter.

Negative initial layer horizontal expansion

This is a feature in Cura and may be called something different in other slicers. The initial layer horizontal expansion can cause the first layer to have a thicker or thinner expansion. Having a thick expansion can help with bed adhesion, but will increase your elephant foot issues. Setting this to a negative number can help to mitigate elephant foot on parts you are having a lot of problems with.

Summary of Fixes and Precautions

- Make sure your nozzle is not too close to the build plate by referring to the “Z-Height Calibration” chapter.
- Confirm you are running your build plate within the suggested temperature range for the material used.
- Utilize a raft to mitigate the problem entirely. Raft information and settings can be found in the “Settings Issues Chapter”.

Extruder Motor Skipping (extruder making a clicking noise)

If you are using a non-geared extruder, especially one set up in a Bowden fashion – such as a stock Ender 3, it is likely you will eventually run into extruder motor skips, where you hear a clicking noise coming out of your extruder. When you look at the extruder, you will see the hobbed gear skip and you will either under extrude, have uneven extrusion, or just not extrude anything.

The most common reason for this is the extruder stepper does not have enough torque to overcome the amount of force needed to push filament through your nozzle. This can happen for a few different reasons, but this is why extruders with gear ratios are preferred. This mechanical advantage causes the stepper to spin faster (higher E-step number), but it reduces the amount of torque put on the stepper motor.

Is your first layer too close to the build plate?

This is covered extensively in the “Z-Height Calibration” chapter, but getting the proper z-height for your first layer is critical. When your nozzle is too far from the build plate, you can be left with a spaghetti monster or clogged hotend. When your hotend is too close, you can damage your build plate or nozzle, or experience extruder motor skips.

When the nozzle is too close, your extruder is trying to push out filament through the nozzle but there is no room for it to escape, so it causes the extruder motor to be unable to push any more material. This will either cause the filament to be stripped or to have the extruder motor skip.

Make sure you give enough room for the filament to lay down properly on the first layer in order to avoid this.

Slow your prints down

A common reason your extruder might be making a clicking noise is that you are running your prints too fast. Your nozzle can only push out so much filament depending on its diameter. So, just as with bottlenecking in traffic, you will experience stoppage if you try to push too fast (especially on non-geared extruders and small nozzle diameters).

Printing too quickly can result in grinding of your filament or extruder stepper skipping. The general rule of thumb is to not print faster than 100x the nozzle diameter on non-geared extruders. So if you are using a 0.4mm nozzle, you should limit your print speeds to 40mm/s, and adjust according to your performance. This may be slow to some experienced people in the industry, but is the rough estimate I use for printing on a non-geared, stock extruder. I run closer to 60mm/s on a good geared extruder, and you can go even faster if your frame allows for it.

You can test this out mid-print if you have an LCD screen on your machine. Most LCD setups are designed so that when you turn the knob mid-print, it will change the feed rate (speed). Newer machines will have something you can touch or go to that is called “Feed Rate” or “FR”. If you hear clicking and would like to see if reducing the speed can fix the issue, reduce the feed rate. Set it to 90% or lower to see if the skipping is decreasing. You can also just reduce the speed in the slicer and slice a new G-code.

If you are still seeing this problem, you may want to check if there is too much moisture in your filament, as covered in the “Stripped Filament” chapter.

Increase the extrusion temperature

Before attempting this, make sure that your issue isn't being caused by heat creep (refer to the "Nozzle Clogs" chapter of this book). If you are experiencing a clog in your barrel due to heat creep, increasing your extrusion temperature will only make the problem worse.

If you are not experiencing any heat creep and the barrel of your machine remains close to room temperature, you can try increasing the extrusion temperature a bit to decrease the chances of your stepper motor skipping. You will normally not want to go outside the recommended print temperatures, but there have been a few times I have had to do this to print properly. This may not work, but if you print at a lower temperature than is needed for your filament, it won't get to the proper viscosity.

This increase in temperature, so long as you are still within the material's accepted extrusion temperature range, will allow more filament to feed through the nozzle at a faster rate.

Be careful when swapping material types

Just as with nozzle clogs and having built up material on the nozzle, you want to be careful when swapping filament types in order to avoid extruder motor skips. When you swap from a higher temp material to a lower temp material, such as from ABS to PLA, you need to make sure you clear out all of the previous material before continuing. If you just heat your hotend to 210°C and push PLA through, it is likely that there is still some ABS left inside.

Please see the “Built up Material on Nozzle” for a full explanation on how to swap materials, but the best bet is to do a cold pull, where you push filament through at the temperature of the filament previously used, then let the hotend cool to 130°C, and pull the filament out. Repeat until all the previous material is gone.

Having any debris in your hotend, or anything that can cause clogs or gets in the way of the filament path, can lead to extruder motor skips. Be sure to read the chapter “Nozzle Clogs” for further ways to prevent this from happening.

Too small of a nozzle

The smaller the nozzle you use, the more likely you will experience stepper motor skipping. I had a near impossible time printing with a 0.25mm nozzle and a non-geared extruder. Minor motor skips led to a print looking as though it was grossly under-extruded.

This is due to an increased bottlenecking effect at the point of the nozzle. I was forced to upgrade to a geared extruder to print with these finer nozzle diameters. E3D actually advertises that their lower diameter nozzles require a geared, direct extruder to work.

Loosen the tension on your idler

Most extruder setups have an idler that allows for you to adjust tension by pinching your filament against the hobbed gear or bolt. This tension is necessary to prevent filament grinding and to make sure the proper amount of material is being pushed through the extruder.

While a decent amount of tension is required, you can of course go too far and have this idler be too tight. When too tight you can actually flatten the filament, making it too wide to feed. When material is too wide to feed you will experience similar issues as you would with heat creep. You may also experience stripped filament, but it can also result in the skipping of your extruder motor.

Pinching too tight on a motor that does not have a lot of torque can also cause skipping at the point of contact. While a tight idler allows for good grip on your filament, it is harder for the extruder motor to spin, especially on non-gear setups.

If you notice that the tension on your idler is very tight and you are experiencing skipping of your stepper, try loosening it a bit. Unfortunately the crummy extruders that come stock on inexpensive machines like the Ender 3 do not have a way to adjust this tension. The only way to reduce the idler tension would be to cut the spring, or use a different one, meaning you would be in jeopardy of making the idler too loose. If it becomes too loose, you can experience under-extrusion as the hobbed gear or bolt will begin to slip on the filament. Just one other reason why I do not like those extruders.

Making sure filament path is clear

The first step in making sure your filament path is clear is to check for nozzle clogs and residue in your hotend setup. You can do this by torching out the old plastic in a well-ventilated area.

Aside from old material and debris, be sure to check that the actual pathway that your filament is traveling before being fed through the extruder is clear. If you have a 3D printed carriage that is warped, or one that is not designed properly, you may have a pathway that does not allow your filament to pass smoothly through it. Any big turns that are required to get your filament to go down your barrel will add to the difficulty involved with feeding material. Resistance at the spool or obstructions in the pathway leading to the extruder will also cause problems.

You may need to print or purchase parts for a new extruder with tighter tolerances and a clearer path to the hotend. This is yet another reason you only want to buy hotends from reputable manufacturers that have tight tolerances.

All-metal hotends allow for heating without the need of Teflon tubing. This Teflon tubing can become deformed over the course of a lot of heating, making the filament path obstructed. I always suggest using an all-metal hotend (such as E3D or Microswiss) whenever possible.

You should also disassemble your extruder and make sure nothing is blocking that passage either. There could be a piece of broken filament in there that is blocking material from passing smoothly.

Push PTFE tube all the way down

If you are using a stock hotend that isn't all-metal, you will want to make sure your PTFE tube is pushed all the way down to the heaterblock. If there is a gap between your heaterblock and your tube, this will surely cause at least a minor clog, and make your extruder skips worse.

Check your extruder stepper VREF

If you are running a board with digital current control, when you flash firmware onto your printer there is a section that tells the board how much current is flowing to the steppers. You will essentially want this current to be the lowest possible that still gets the job done. This is because setting the current too high can cause overheating, the driver to run less efficiently, add wear on the stepper motor, and result in an increase in noise.

Other boards have a manual current control, including the one that comes on the common Ender 3V2. For these you will need a multimeter and a screwdriver. It is smart to switch to a geared extruder before doing this, but it is worth checking if you have a lot of extruder motor skips and can't figure out why. This explanation is elsewhere in this book, but it is relevant to underpowered stepper motors.

The current being improperly set should not be an issue with factory made machines, but will be more common on the inexpensive DIY ones. That said, I have had to change the stepper driver on one factory made machine that was experiencing an overheating issue due to the current being set too high. In fact, there are quite a few stories I hear of VREF settings being incorrect on factory made Creality machines, so this is definitely worth checking if your extruder is having constant skips and you have tried other methods.

In order to check and change your VREF, you first want to have your machine turned off and the stepper motor cable disconnected from the board. You then need to look up both the stepper motor you are using, as well as the stepper driver, or just search online for VREF values on your particular printer.

Current limits are determined in the motor and driver data sheet. You will not want to run higher than either the driver continuous current limit, or the motor current rating limit, so it is often good to have a driver that has a higher continuous current rating. I suggest going off of the continuous current limit of the motor. If this is set too low, your motor in question will skip easily.

Once you know what current limit you want, you then need to find out the calculation for your stepper driver to determine a VREF. You can go to the current limiting section of your stepper driver data sheet in order to figure out the VREF you will want. Current limit will equal $VREF \times 2.5$ for standard A4988, and $VREF \times 2$ for DRV8825. There is then another option by TMC where the calculation is slightly different, where $VREF = (\text{Motor current} \times 2.5) / 1.77$. To find out your ideal VREF, there is a handy calculator online called "Stepper Driver VREF Calculator", where you enter the rated current

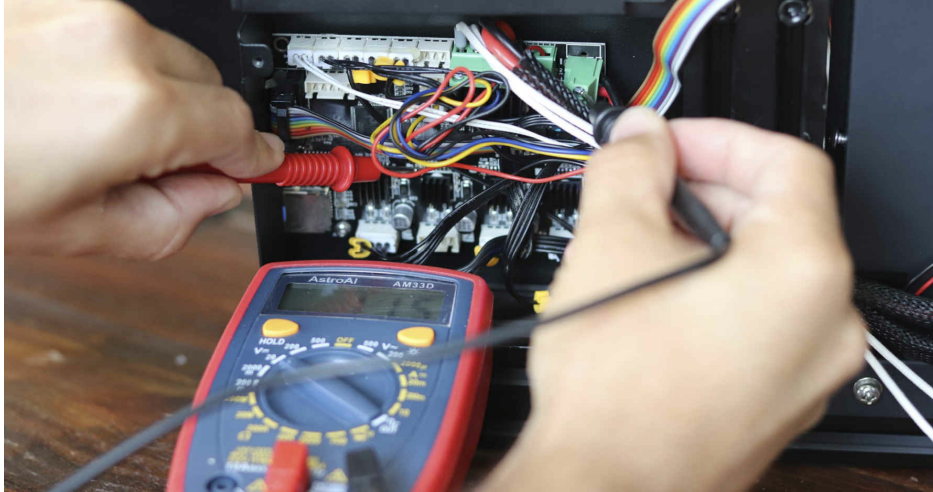
of your motor and it will read out the ideal Vref numbers for each driver type. Even with all of these tools it might still be difficult for you to figure out exactly what your machine needs, so searching online for your specific printer and its recommended VREF or voltage is ideal. This helps a lot since many people have already figured out proper settings for popular machines. Going off of TH3D's website, an Ender 3 V2 should have a VREF in Volts for the extruder motor at 1.35 – 1.4V.

I understand this is very confusing for someone who is new to this, just as it was for me. But essentially the VREF is the power that is being sent to your motors and it can be tweaked depending on your motor and stepper driver setup. So if your max current limit on your motor is 1AMP, and you are using the standard A4988 driver, you will have a VREF of roughly 0.4. This is because $0.4 \times 2.5 = 1$. This 0.4 would be your target number that your multi meter will read out. If your VREF is lower than 0.4, you will not have enough power sent to the motors, resulting in layer shifts and motor skips. Going higher than 0.4 can result in an overheated motor.

When dealing with the Ender 3V2, you want your multimeter to read out a number between 1.35V and 1.4V for the extruder motor. If you test out your Ender 3 V2 extruder VREF and you get a reading under 1V, it is likely a reason your extruder motor skips.

To test and change your VREF, you will need to plug back in your power and turn the machine on, while you still have access to the board. Be careful now that everything is on. Make sure the driver you are testing has the stepper motor unplugged from the board. Then grab your multimeter.

Set your multimeter to 20V DC, and touch your black negative lead to a ground pin on the stepper driver (titled GND). If you are unaware of which pin the ground is, you can also touch the black negative lead to the negative section of your power supply. On my Ender 3 V2, I just touch the metal on the screw terminal where the black cable is going from my board to my power supply. Just make sure to only touch a ground section with your black negative lead.



You can then clip the positive lead of your meter to the metal shaft on the screwdriver to help read everything out while you change it. If you do not have a clip to connect the lead to the screwdriver, you will need to test, tweak, and test again. Then touch the positive lead (or the screwdriver if you have it clipped) to the potentiometer on the stepper driver. This is a very tiny screw-like object on the driver. You will then see a voltage number on your multimeter. Remember that your printer is powered on so make sure to only touch a ground and your potentiometer. The number your multimeter reads is your VREF. Be sure your VREF is the proper rating for your machine.

You can turn the screwdriver clockwise to increase the voltage, and counterclockwise to decrease it (which is actually the opposite on some drivers, so just make sure you are testing after each small turn). A 1/8 turn of the potentiometer will make a drastic difference in your VREF, so make sure to not turn too fast. Turn a very small amount, retest, and then continue until your multimeter is reading the correct voltage.

Once you get the proper readout for your printer, plug everything back in and run another test print. If your extruder VREF was set too low, then this should drastically reduce extruder motor skips.

Sometimes you will have to reduce the VREF on your extruder stepper motor if you are swapping from a full-sized stepper to a pancake one, such as going from a standard Ender 3V2 extruder to something like the BIQU H2, which uses a smaller stepper motor to turn the extruder.

Upgrade to an extruder with a proper gear ratio

I absolutely hate the extruders that come stock on most inexpensive machines, since they do not have a proper gear ratio. This means that the extruder has a 1:1 ratio, which means it is giving your motor no mechanical advantage. Each turn of the stepper motor has a direct 1:1 relationship with how much your toothed gear turns which pushes out your filament. Even the metal dual-drive extruders that some Creality machines have do not have any mechanical advantage, and will still lead to extruder motor skips.

If you are printing with a very small nozzle, or attempting to print quickly, you will almost certainly need an extruder with a gear ratio. Something like the Hemera, my favorite extruder/hotend combo, has a 3:1 gear ratio. Along with the great dual-drive grip and non-existent distance between the extruder and heat break, this extruder adds a mechanical advantage which means much less stress is put on your stepper motor. When using the Hemera I have had zero extruder motor skips.

You don't specifically need a Hemera; you can get any extruder with a gear ratio and it will help fix this problem. Bondtech, BIQU, DropEffect, and many other companies make great extruders with a gear ratio, and many of them have a dual-drive feature as well.

When you do swap to a geared extruder, you will need to change your E-steps. With a higher gear ratio, your stepper motor has to turn more in order to extrude the same amount of material. Find the starting point for your extruder by searching online. The Hemera has a good starting point around 400 for E-steps. From here you will want to hone in the exact E-steps. This is covered in the "Over and Under Extrusion" chapter.

Upgrade hotend if using a very large nozzle diameter

When you are using a very large nozzle diameter, something like 1mm, it is going to be difficult for your material to reach the proper viscosity unless you print very slowly. This is because you are now pushing a lot more material through the nozzle. If you want to print at decent printing speeds with these larger nozzle diameters, you will want something like the Volcano hotend. I cover this more in the “Speed Limitations” chapter.

Summary of Fixes and Precautions

- Confirm your first layer isn't too close to the build plate.
- Slow your prints down (manually change during print with the knob on the LCD screen to see if this fixes the problem).
- Check to see if there is too much moisture in your filament by drying it, or swapping to a new spool.
- Print at a slightly higher extrusion temperature.
- Loosen the tension on the extruder idler.
- Have a clear, straight filament path. Reprint or find a more suitable extruder carriage on Thingiverse if required. Disassemble both the hotend and extruder to see if anything is blocking it from moving.
- Increase the current to your stepper if not at limit and not experiencing overheating. Too little power to your stepper will definitely result in skips.
- Increase nozzle diameter if using smaller than a 0.4mm nozzle and printing without an upgraded extruder.
- Upgrade to an extruder with an increased gear ratio. The crummy stock ones that come on inexpensive machines just don't get the job done. My favorite is the Hemera, but you can go with competitors such as Bondtech, BIQU, DropEffect, and so many others. Even an old Titan extruder by E3D will drastically reduce any stepper skipping issues, since it has a proper gear ratio.
- If you don't want to print at a crawl and you are using a larger diameter nozzle, you will likely want to upgrade your hotend. Large nozzle diameters require something like the Volcano hotend or a similar competitor.

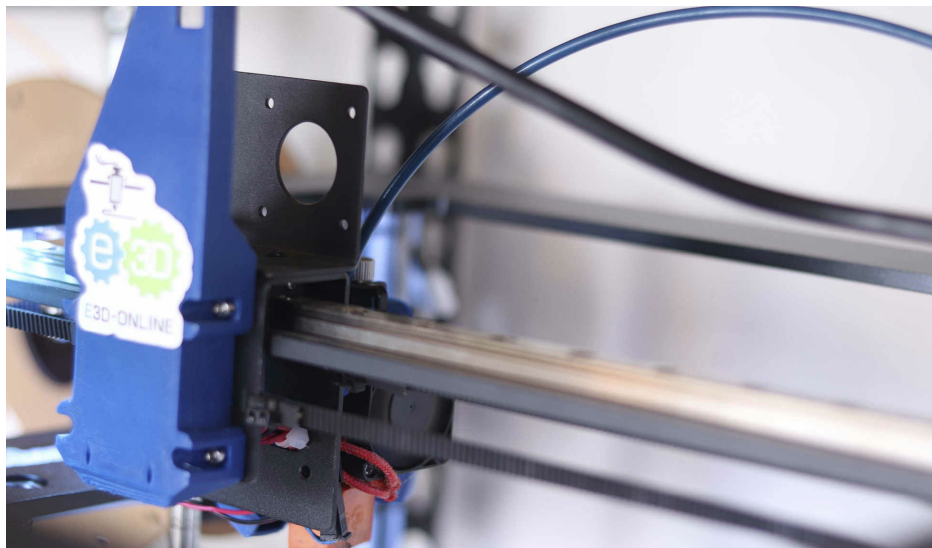
Filament Snaps

This issue can be extremely frustrating since it can happen hours into a print. This chapter's explanations are very similar to previous editions of this book.

Understand the material you are using

Certain materials have lower elongation than others, meaning they will snap easier on the spool. Almost all of the carbon fiber blends snap extremely easily on the spool and will require you to use Teflon/PTFE tubing to guide the filament and help prevent breaks mid-print.

Using Teflon tubing to guide your filament to the extruder will help with filament breaks and tangles. In the photo below I have a PTFE tube going from my spool to my Hemera extruder, which helps prevent filament snaps and snags.



PLA has a much lower elongation than ABS, so you will experience more frequent snaps when printing with it. You may have noticed this yourself since you're able to snap a piece of PLA off of its spool, while ABS will require a tool to cut it.

3.00mm filament is also far more likely to experience snapping than 1.75mm filament, as explored in the "Upgrades and Purchasing a New Printer" chapter. The larger the diameter of the filament, the more likely it will be to snap. For example, 3mm (or rather 2.85mm) PLA near the end of the spool can be impossible to print without snapping occurring. It seems that the vast majority of printer manufacturers have standardized to 1.75mm over the past couple of years.

Properly store your filament

Not storing your filament properly is the most common reason for filament snapping when it shouldn't. If you do not plan on using a spool of filament for a week or longer, you should store it in a controlled environment.

You can store opened filament in an enclosed environment with a dehumidifier, or vacuum seal it with a desiccant bag. The dehumidifier should be set as low as possible (normally around 20%), especially if you live in a humid area.

This is extremely important when using nylon filaments because they can easily absorb moisture within hours of being exposed to a high humidity environment. Dehumidifying boxes are essential to use when printing with nylon.

I currently use an EIBOS drying box for any spool that has been out too long or hasn't been used in a while in order to bring its moisture back to where it should be. You do not need to buy an EIBOS; many companies make drying options. I cover this drying process more in the "Materials and their Settings" and the "Stripped Filament" chapter.

Change manufacturer

If the manufacturer of your filament does not have high reviews, you may experience frequent breaks. This is when spending a bit more money for a name brand can pay for itself. Hatchbox, E-Sun, Polymaker, Matterhackers, ColorFabb, IC3D, taulman3D, AIO Robotics, Overture, Prusament, and Proto-Pasta are just a few of the filament manufacturers that should have no issues. There are quite a few others that I have not tried, so it is worth reading reviews.

Loosen idler tension

The tension on your extruder idler may be pinching your filament too tight. If this is the case, the grinding of filament may lead to your material snapping entirely.

Most extruder setups allow for the adjustment of this tension. You want it to be tight enough for there to be plenty of pressure, but not too tight that it grinds into the filament or deforms it. If you think this may be too tight, see if loosening the idler fixes your problem.

Check for nozzle clog

You can be experiencing a nozzle clog if your filament is snapping, so if this is the case, be sure to read the “Nozzle Clogs” chapter in this book.

Summary of Fixes and Precautions

- Be aware that the specific material you are using might break.
- Add PTFE tubing to help guide the filament from your spool.
- Switch to a 1.75mm extruder and hotend setup for less frequent filament snaps.
- Properly store your filament. Using old or improperly stored material is the most common reason for filament snaps.
- Always buy from a well-respected manufacturer for 3D printing filament.
- Replace your spool and try a new filament manufacturer if needed.
- Check for a nozzle clog and fix it.

Gaps in Walls



This particular issue is something I have personally noticed to happen less frequently on Cura, but it could happen on just about any slicer. This is when the walls of your print do not connect and can result in gaps.

Why it's happening

If your walls are not designed to be a multiple of the thickness of your layer lines, you will have gaps in your walls. Let's say your walls are 1.8mm in thickness, and you have your layer lines set to 0.4mm. Your slicer will either create 2.0mm walls (5 shells), or it will err on the lower side and create a 1.6mm wall (4 shells), causing a 0.2mm gap. I have noticed that all slicers go with the latter, as to avoid printing a part that is too thick.

This gap is normally filled via your infill percentage or by top layers, but you may run into issues when it doesn't.

Each slicer seems to deal with this in a different way. Most will bunch the shells together on each side of the print, with a 0.2mm gap in the center. I believe Slic3r spreads each wall out with a smaller gap, as shown in the photo above, though I may be wrong. I personally do not use Slic3r, but the photo above is from someone who purchased the 2019 edition of my book using Slic3r who had this issue.

If you want your shell walls to fill the entire area of your print, you need the walls to be a multiple of your layer line width. If not, you can try the following solution.

Fill Gaps Between Walls



As you can see by the description, this setting is meant to fix this exact problem. If the gap between walls is thinner than your layer line width, this should fill in that area. Simplify3D should have a similar feature called “gap fill”. You will want this set to “everywhere”, which should now be the stock setting on Cura.

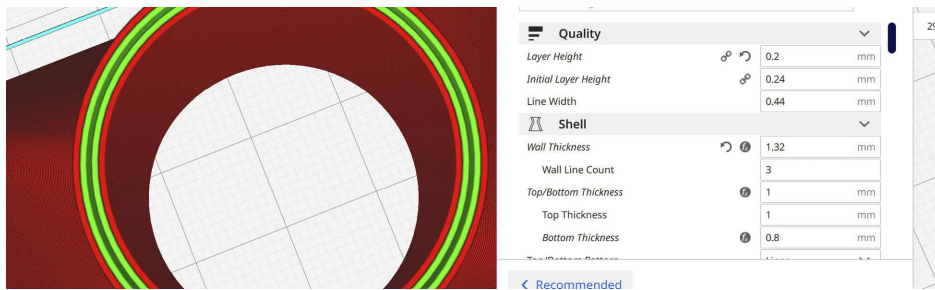
This doesn’t always do the job, but when it doesn’t, normally your infill percentage will.

This setting may cause blobs to appear on the side of your print, so if you are experiencing blobs, you can choose to “Filter Out Tiny Gaps” which is also in the “Walls” section in Cura.

Make your walls a multiple of your line width

The ideal solution would be to redesign the wall in question to be a multiple of your desired line width. If this isn't possible for you, you can slightly tweak your line width to work.

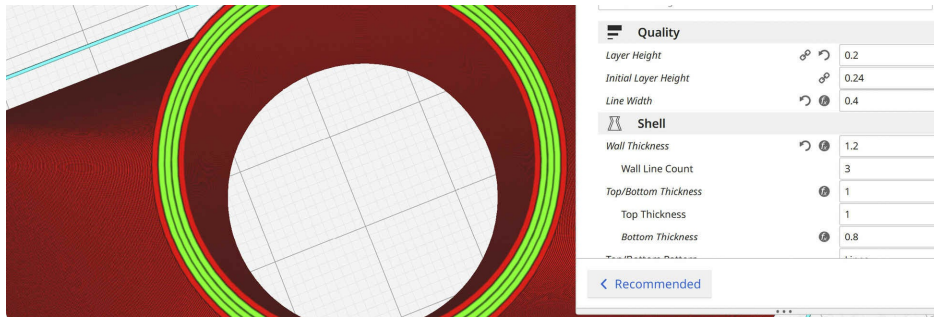
As with the example in the beginning of this chapter, if you need your shells to fill up a 1.8mm wall, using a 0.4mm line width will cause a 0.2mm gap. You can fix this issue by upping your line width to 0.45mm instead. You will then have 4 lines at 0.45mm, which will be exactly 1.8mm without any gaps being created.



If you go into the “preview” section of your slicer after slicing, and you see there is a minor gap in your walls, you can adjust your line width accordingly. While I normally prefer to keep my line width the same as the nozzle diameter I am using, you can go up to 120% of the nozzle diameter and not lose much quality, if at all.

The example above has the line width to 0.44mm and it can only fit two shells (two inner, two outer – making 4 walls), so there is a gap in the middle of these walls. This part is 2.0mm thick, so you can see why this is happening. This is even with the “Fill Gaps Between Walls” setting checked. This goes away if you have infill % on, but the above example is with 0% infill. If you require the line width you have selected, then you will need to have infill fill in that gap.

You can now see what this same print looks like with 0.4mm line width and no infill:



You can see how the gap disappeared, since at 0.4mm line widths the printer is able to fit another wall. You then have 5 walls making 2.0mm; the exact thickness of the part.

You can actually just change the line width of the walls if you choose, so that it doesn't affect the rest of your print. Cura allows you to tweak the wall line width, or even just the inner or outer wall line width – which is located in the “Quality” section. This means for the example above, you can have your wall line widths set to 0.45mm, and the rest of your print be 0.4mm. You could also set your outer wall to 0.42mm and then your inner walls to 0.46mm, which would result in the same 1.8mm with 4 shells.

Reduce your Top/Bottom Line Width

Though the best method would be that described above, you can also just reduce your top layer line width as to allow for the best chance of filling in the gaps as much as possible. If you go into Cura and change the “Top/Bottom Line Width” located in the “Quality” section, you can reduce it as low as possible for your nozzle diameter. While I prefer to keep my line width near the same as my nozzle diameter, when it is just for your top layer it is not as important. You can go all the way down to about 0.26mm on a 0.4mm nozzle and still have a high quality top layer. Setting a thinner line width will allow for less gaps between your walls.

Potential under extrusion

When the walls are not touching at all, similar to the photo at the beginning of the chapter, you could be under-extruding. As explained by Ultimaker on one of their guides for Cura:

“If the walls are not touching each other at all it is an extrusion issue. Cura is asking your printer to create a series of 0.4mm lines and is spacing them so that they fuse together. However, if your printer is under-extruding slightly, the lines will be marginally thinner and they no longer fuse together properly. The solution could be as simple as reducing your print speed slightly or increasing your temperature a few degrees.”

If reducing print speed or increasing your temperature does not fix the issue, you can check the “Over and Under Extrusion” chapter for further information.

Gaps between infill and walls

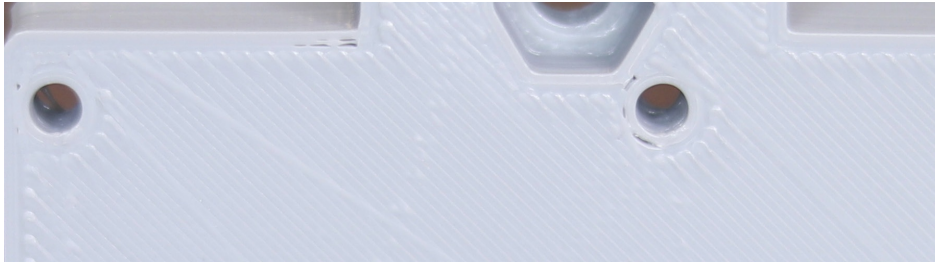
This is a different issue than the one explained previously in this chapter. The most common reason for having a gap between your infill (or top/bottom layers) and the walls, is the infill overlap percentage being set too low. I normally will keep my infill overlap percentage around 7 or 8%, but some people suggest going up to 30%.

This could also be because your “Skin Overlap Percentage” is set too low. Cura has this set to only 5% stock, but you can bump it all the way up to 50% without much issues. Refer to “Gaps on Top Layers” for more information on this particular issue.

Summary of Fixes and Precautions

- Choose “Fill Gaps Between Walls” on your slicer, and make sure you have your infill % set higher than 0.
- Attempt to make your walls a multiple of your line width. If not possible, lower your wall line width to as low as possible.
- Check to see if you are under-extruding
- Slow down the speeds of your shell walls.
- Check your infill overlap percentage when dealing with gaps between your main part and the shell walls.

Gaps on Top Layers



This is an issue I am sure many of you experience without even recognizing it. The YouTube channel called “CHEP” has a video titled “Cura 4.8 Slicer Tips for Eliminating Gaps in your 3D Prints” which covers this exact issue.

Top/Bottom Line Direction and Top/Bottom Line Width

On Cura there are two areas you should inspect first. First is the Top/Bottom line direction. Cura has this set to 45 degrees, which is fine for most prints. If you are experiencing gaps on the top of your print, the first thing you should do is set this to 90 degrees instead. This change will help to reduce these gaps. This is located within the “Top/Bottom” section on Cura.

The second thing you should look at is the “Top/Bottom Line Width.” This is more important than the degree angle, and has to do with a similar issue as I covered in “Gaps in Walls.” If the surface area that needs to be covered on the top of your print is not a perfect multiple of your line width, it is inevitable you will have some gaps. Sometimes they are so small you won’t notice, but other times it will cause the issue that lead to me writing this new chapter. This will be found in the “Quality” section on Cura.

Quality		
Layer Height		0.2 mm
Initial Layer Height		0.3 mm
Line Width		0.4 mm
Top/Bottom Line Width		0.26 mm

The best way to fix this would be to reduce your Top/Bottom Line Width to the lowest possible for the nozzle diameter that you are using. For a 0.4mm nozzle, that would mean about 0.26mm. I wouldn’t suggest going this low for the entirety of your print, just for your top layers. This means that you are far more likely to fill in all of those holes even if the top of your print isn’t a perfect multiple of 0.26mm. The lower you can reduce this number, the better your odds are of reducing gaps. This is definitely the most important factor when it comes to reducing gaps.

Skin Overlap Percentage

 Top/Bottom 		
Top/Bottom Thickness	0.9	mm
Top Thickness	0.9	mm
Bottom Thickness	0.9	mm
Monotonic Top/Bottom Order		
Top/Bottom Line Directions	 45	
Enable Ironing	☐	
Skin Overlap Percentage	 50	%

Another setting that will help reduce gaps, particularly the gaps between your top layers and the walls, is the "Skin Overlap Percentage," also found in the "Top/Bottom" section on Cura. Just like infill overlap, this is the amount those layers will overlap with the walls on your top layers. Cura has this set to 5% stock, which is too low. You can bump this all the way up to 50% and reduce any of these gaps between your walls and your prints.

Fill Gaps Everywhere

This is also covered in the “Gaps in Walls” chapter, but worth mentioning again. I have this “Fill Gaps Between Walls” set to “Everywhere” which is found in the “Walls” section on Cura. This really helps to fill in those tiny gaps that may be occurring that aren’t solved via using the previously described methods.

Summary of Fixes and Precautions

- Change the “Top/Bottom Line Direction” to 90 degrees from Cura’s set 45.
- Change the “Top/Bottom Line Width” to the smallest you can with the particular nozzle you are using. That is about 0.26mm for a standard 0.4mm nozzle.
- Increase your “Skin Overlap Percentage.” Stock Cura is set to only 5%. Bump this up a decent amount, you can go up to 50% without any real issues.
- Change “Fill Gaps Between Walls” to “Everywhere.”

Ghosting

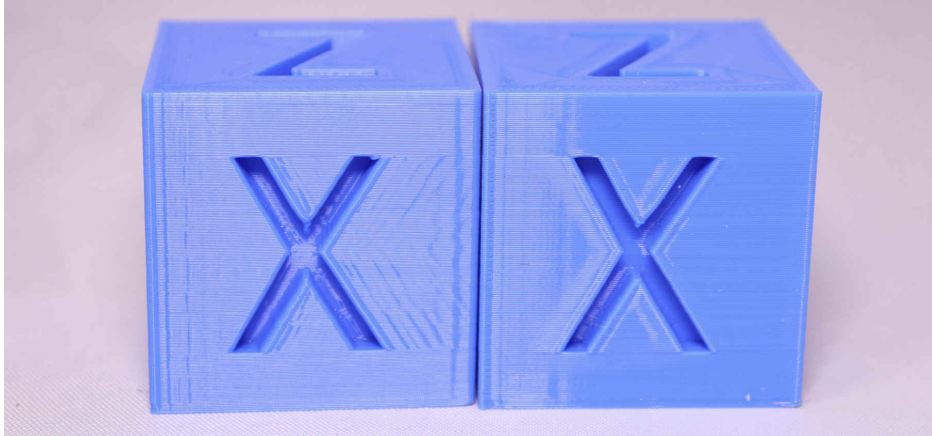
Ghosting refers to an “echo” where details in your print can be seen outside of where they should be. This is sometimes also called “ringing”.

The main reason ghosting occurs is having your acceleration and jerk settings too high. This is extremely common since many printer manufacturers will auto set the default value for these numbers too high because it allows them to advertise faster printing times.

A part with some minor ghosting is still entirely useable, so printer manufacturers may get away with calling something a successful print that we as makers would not.

Keep in mind that ghosting should be reduced on a CoreXY or Delta machine, since the increased weight of moving the print bed on Cartesians will make this problem worse.

Reducing jerk and acceleration



As you can see from the photo above, there is a drastic difference in ghosting between the two examples. These two parts were printed with the same slicer settings, except for Jerk and Acceleration.

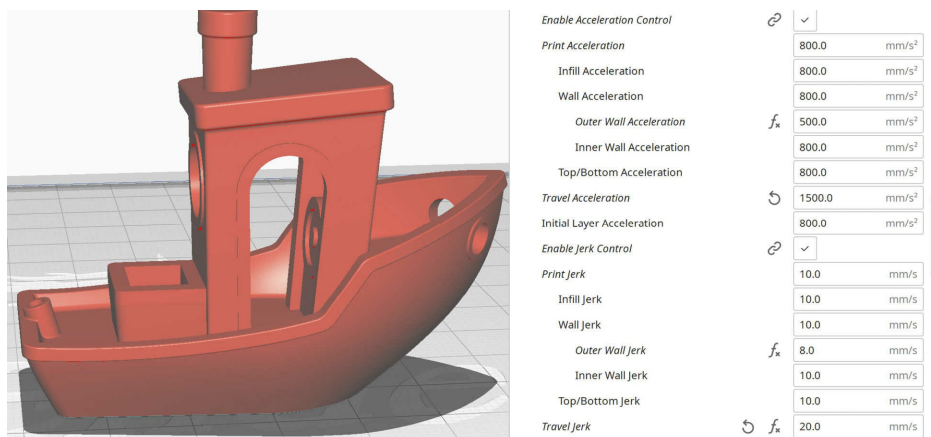
Acceleration is pretty self-explanatory, but jerk refers to the initial speed after a directional change. After stopping, your print head will start at your jerk speed before then accelerating to your print speed.

The failed print on the left had accelerations of $3,000 \text{ m/s}^2$ and a jerk of 30 mm/s while the print on the right had accelerations of 500 m/s^2 and a jerk of 12 mm/s . It has become clear to me that reducing jerk and acceleration are the key factors when it comes to reducing ghosting.

In the photo above, the print on the left took 1 hour 36 minutes and the print on the right took 1 hour 50 minutes. Therefore, reducing your acceleration and jerk will require more time for your print to finish, but it is definitely worth it to have the quality you expect.

Controlling jerk and acceleration in Cura

As mentioned elsewhere in this book, the newer versions of the Cura slicing software is updated frequently and now allows for jerk and acceleration control.



This allows you to change these numbers without having flash your firmware. In Cura, this is located under the “Speed” section. If you prefer not to use Cura, then you will have to check within your slicer to see if this option is available. If it isn’t, you will have to manually change the jerk and acceleration within Marlin and re-flash your firmware. Almost all slicers now allow for acceleration and jerk controls, so this flashing of firmware shouldn’t be needed anymore. Some printers even have the ability to change your top acceleration and jerk settings on the LCD screen.

Add small cushions under printer feet

This is a minor but easy addition to your printer to help reduce ghosting. Part of the problem with ghosting is the rattling that occurs within the printer without anywhere to disperse vibrations. If you have a very sturdy printer on a sturdy print area, most of the rattling ends up happening within the machine.

An easy fix is to grab some small foam cubes and place them under your printer feet. This will help to disperse vibrations through the machine into the foam, allowing you to print higher acceleration and jerk speeds with reduced ghosting.

While I have these pads on my printer, I still do not print faster than roughly 800m/s^2 acceleration and 15mm/s jerk on my Cartesian setup. Remember that CoreXY machines can have much higher acceleration and jerk settings without an increase in ghosting. Delta machines can go even higher than CoreXY due to how the X and Y axis are moved.

Having a lighter carriage

The lighter the carriage, the less ghosting you will experience. This means ghosting will be more of an issue on a direct extruder than a Bowden, due to the increased weight of adding a stepper motor. I will still always prefer direct extruders to Bowden due to the extra amount of materials available to print, but one of the benefits to Bowden is the reduced weight.

You can also reduce the weight on your carriage via a smaller/lighter extruder stepper and/or carriage. The Hemera hotend/extruder is a great setup, but if you own one you will notice just how heavy it is. This weight might be too much for your printer, meaning you really need to reduce your acceleration and jerk to remove ghosting.

Something like the BIQU H2 is similar to the Hemera but has a pancake light stepper, resulting in a drastic reduction in weight. The same is true for the OmniaDrop by DropEffect that I reviewed, which has a much lighter stepper motor than the Hemera.

Belts too tight

While belts being too loose can cause Z-wobble or layer shifts, belts being too tight can cause ghosting. There is no specific standard for how tight a belt should be, but I generally have them tight enough to not droop and still be springy to the touch. If they are too tight it can stretch out the belt and cause a reduction in dampening, which adds to ghosting problems.

Having too tight of a frame

You almost always want to have a well-built, strong frame. Unfortunately, if you do not have any dampeners for your axis, this means that the vibrations from your machine won't be dispersed, resulting in increased ghosting.

If your frame is too loose it can result in Z-wobble. The key is to find a happy medium where you have no Z-wobble and no ghosting; something fairly hard to achieve and a reason CoreXY is preferred over Cartesian.

Please refer to my YouTube video titled "Reduce Ghosting in 3D Printing" for further information.

Summary of Fixes and Precautions

- Reduce your acceleration and jerk via Cura or your preferred slicing software.
- If not available in your slicing software or on your LCD screen, reduce via marlin and re-flash.
- Add small foam cushions below your printer feet to help disperse vibrations.
- Reduce weight on carriage if possible.
- Have dampening for your axis if possible.
- These problems are less of an issue on CoreXY and Delta machines

Hotend Can't Reach or Maintain a Temperature

This problem can arise in a couple different fashions. Your printer may be able to reach its set point but cannot maintain it during the middle of the print, or your hotend may not be able to reach the temperature you give it at all.

Confirm you are using the proper wattage heater

I had a consistent issue on one machine where it had extreme difficulty maintaining 230°C, and could never reach 250°C. It turned out I was using a 30W heater instead of 40W, which the machine was rated for (Lulzbot TAZ 5 in this example).

Replacing the heater with the proper wattage fixed this problem instantly. Wiring for the heaters is typically color coded by wattage, so make sure the colors match and the rating on the cartridge are the same when changing out your heater.

Check firmware for MAXTEMP

It is also smart to confirm that the maximum temperature for your extruder is set higher than you will be printing in your firmware. Your printer will be restricted from reaching any temperature higher than this.

```
// When temperature exceeds max temp, your heater will be switched off.  
// This feature exists to protect your hotend from overheating accidentally, but *NOT* from thermistor short/failure!  
// You should use MINTEMP for thermistor short/failure protection.  
#define HEATER_0_MAXTEMP 270  
#define HEATER_1_MAXTEMP 245  
#define HEATER_2_MAXTEMP 245  
#define BED_MAXTEMP 200
```

This should be automatically set to a safe number for your machine and hotend, but if it was ever turned to 230, you would not be able to reach any temperatures above that. If you were to upgrade to a high temp hotend, you would likely need to tweak this number to what the new hotend is rated for. Don't set this to a high number if you are unsure of your hotend setup. Hotends that are not all-metal should not be printed above 250 degrees.

PID autotuning

PID refers to a proportional integral derivative controller which is used to control the temperature of the nozzle. When the PID settings are not correct, something as minor as your active cooling fan can drastically change the temperature of the hotend.

An active cooling fan, or even the barrel fan can drop a hotend from 210°C to 160°C if the PID settings are off. This rapid cooling will make the nozzle too cold for the filament to feed, or may even prevent your print from starting at all.

If the temperature is fluctuating consistently around the set point that is a sign that the PID settings need to be tuned. Erratic fluctuation, however, is more likely to be a sign of intermittent electrical failure or the temperature sensor being dislodged from its cavity. So definitely make sure your thermistor and heater are connected properly to the hotend as well.

The process of setting new PID settings involves running a test which is called a PID autotune. This will then tell you what your proper PID settings should be.

First step in running a PID autotune is to mimic the material you will be printing with (or at least the material you are having difficulty with). If you will be using an active cooling fan, manually turn it on. While the hotend is still at room temperature, give the G-code command below through your printer terminal (Repetier if hard wired to your computer, Octoprint if remotely controlling).

M303 E0 S[temperature]

So if you are testing out PLA, and you want to print the PLA at 200°C, you would type the G-code command:

M303 E0 S200

This will run a PID Autotune for 200°C. After a decent amount of time your program will feedback values for P, I and D as shown below:

```
bias: 92 d: 92 min: 196.56 max: 203.75
Ku: 32.59 Tu: 54.92
Classic PID
Kp: 19.56
Ki: 0.71
Kd: 134.26
PID Autotune finished ! Place the Kp, Ki and Kd constants in the configuration.h
```

If your PID Autotune has failed, and you have confirmed you are using the correct wattage heater, skip to the next sections starting with purchasing a new heater.

If it didn't fail, you can then enter these new PID values a few different ways. One method would be to open up marlin and go to your configuration.h tab in order to change the PID settings for your entire machine, as shown in the photo below.

```
// If you are using a preconfigured hotend then you can use one of the value sets by uncommenting it
// Ultimaker
#define DEFAULT_Kp 22.2
#define DEFAULT_Ki 1.08
#define DEFAULT_Kd 114
```

When in the configuration.h tab, scroll until you find the Kp, Ki, and Kd values, as shown above. You would then change these values to what the PID autotune test read out.

Another way to do set your PID values would be to change them on the printer itself. Some machines let you change this within the EEPROM of the LCD screen. Just make sure you store settings if you go this route.

The last and easiest way to set your PID values is to send a G-code command by typing in:

M301 H1 P##.## I##.## D##.##

For example, I would type in “***M301 H1 P19.56 I0.71 D134.26***” and click enter. This would then set your new PID values. After entering this in, type “***M500***” to save your settings.

Get a new heater

This normally isn't needed, but I have had a printer shipped to me that couldn't maintain its temperature straight out of the box. I ran PID autotune a dozen times and the temperature still fluctuated all over the place. It turns out that it was just a malfunctioning heater. Replacing the heater instantly fixed the issue.

Readjust or reprint active cooling fan

The most common reason for hotend temperatures dropping mid- print is that your active cooling fan is blowing directly onto your nozzle. With your hotend at room temperature, turn on your active cooling fan. Then place your finger onto your nozzle and hotend and see if you feel a lot of cool air. If you are, this air may be what is dropping your hotend temperature during your print.

You may be able to overcome this differential by running your PID tuning while the active cooling fan is on, but this doesn't always work. The best active cooling fans are ones that go around your hotend and blow directly downward. You only want the air to be blowing onto your print and not onto your nozzle.

If your active cooling fan is blowing directly onto your hotend, you may want to print a new duct. Search on Thingiverse and elsewhere for a file that will fit your extruder/hotend setup. There are likely dozens already created. Make sure that this duct angles the air downward as to avoid the heater block and nozzle.

These fan ducts can get in the way of your printer finding home, or can even get in the way of raised bed clips, so be careful and confirm your machine can move to all areas of your build plate before starting a print.

Add a silicone sock

As mentioned a few times before, you should add a silicone sock to your heater block/nozzle if the manufacturer makes one. Not only does this silicone sock prevent burnt black spots on your print, it can actually help insulate the hotend. Any air blowing around the heater block will not have as much effect on cooling a nozzle that has a silicone sock around it.

If you have exhausted all of the above options and your PID Autotuning is still failing, you will likely need to re-flash your machine. Buy a new thermistor, rewire it to your board, and flash your machine again. Then attempt to run the PID Autotuning code.

If you have re-flashed your machine and replaced your thermistor, and the PID autotune is still failing, you may require a new board or power supply.

Summary of Fixes and Precautions

- Confirm you are using the proper wattage heater for your machine.
- Check that you're not exceeding the MAXTEMP for your firmware.
- Run the PID Autotune sequence for the temperature you are having difficulty holding, along with turning on any fans that will be running during the print.
- Change the PID in marlin to set the standard for your machine, or just set it via G-Code commands (which is my preferred method).
- Make sure your active cooling fan isn't blowing directly onto your heater block and nozzle. If it is, you will need to adjust it or reprint a new fan duct so that it aims directly onto your print.
- Add a silicone sock if available.
- Replace heater and thermistor.
- Flash firmware.
- If all else fails, replace your board and power supply.
- Follow TH3D Studios on YouTube, since they cover a lot of these board issues.
- Search for your particular printer and see if there are others with the same problem.

Hotend Not Heating

This is easy to diagnose since your extruder will not heat up. If the temperature is reading out properly, then there is an issue with your heater. Many of these instructions are similar to the ones in the “Build Plate Not Heating” chapter in this book.

After you confirm you are using the correct Volt/Amp for your heater/board combination, you can move forward to checking the following issues.

Heater malfunctioning

The heater is the cartridge attached to thick wires (normally red or blue) and is connected to the heater block of your hotend. As with all parts on a printer, this cord can wear out and malfunction. Luckily, replacement heaters are fairly inexpensive.

It is best to have a spare heater available so that you can easily test whether it is your heater or your board that is malfunctioning. All you would need to do is disconnect the heater from the board and connect your replacement heater, and see if it heats up without any issue. If it does, then it may not be worth your time to even figure out what is wrong with your old heater. This is why it is great to have all of your important parts use a connector. With a connector, you do not have to cut wires or rewire entirely to the board if you just want to test a part out.

If you do not have a spare heater, you can start by checking the continuity of the wires from the cartridge to the board using a multimeter. This may be difficult for you if you do not have any connectors or exposed wires that you can access near the cartridge. Even if you do find that one of the wires has a break in it, you will likely need a replacement cord, since it needs to be a thick gauged wire. For a 12V 30A heater, you will need at least a 14 gauge wire.

DO NOT REPLACE A HEATER CORD WITH A NORMAL SMALL-GAUGED WIRE (thinner than 14 gauge); YOU WILL BE SUSCEPTIBLE TO A FIRE AND FURTHER BURNT OUT WIRES.

Burnt out and/or disconnected wires and connectors

This is far more common if you are experiencing issues with your build plate not heating, but can also occur with your extruder heater as well. You should easily be able to see any burnt out connectors on your board with the naked eye, and if you can't, you will be able to once you disconnect the hotend from the board.

When a connector or wire is burnt out you will not get any heating out of your hotend. If there is a wire you can easily see was burnt out in one section, you can often fix this without replacing the entire wire.

Not having continuity may be as simple as noticing that a connector has come loose. However, when there is a partial or intermittent break in the wiring, typically referred to as fraying, this issue can be much more difficult to diagnose. This should only ever be an issue on older machines with many printing hours, and on machines where the wires aren't organized. When this does happen, try to replace the entire wiring harness with a new one if available from the manufacturer. When one section of wiring begins to fray the rest typically will follow shortly thereafter. Make sure to rewire in an organized fashion.

To rewire, start with the printer off and unplugged, cut off the burnt section out of the wire, and then solder the two sections that are not burnt out back together (or use solder seal connectors). Be sure to use the proper shrink wrap because absolutely no metal can be exposed after this process. I mention what I use in the "Important Accessories and Replacements" chapter. Then you will need to confirm that the extruder can reach its farthest point from the board, because you just made the wire shorter than it was.

If your newly soldered wire cannot reach the board at the extruder's furthest point, you will experience layer shifts on large builds.

Board overheating

Just as with the section above, your board overheating is more commonly associated with the build plate not heating, but an overheated board can also result in your hotend not heating.

If your hotend is not heating, you can normally tell if your board is overheating by checking the temperature of your board when the heater cuts out. Be careful though, because if the board is overheating, it will be hot enough to burn your fingers.

I go over a few methods on preventing your board from overheating elsewhere in this book, but if you are experiencing an overheated board, you will need to get 1-3 active fans blowing right onto it. Check to see if the fan that is wired to board is working and has intact blades.

If your board is enclosed with the rest of the machine, the build plate heating will drastically affect the ambient air temperature, causing your board to overheat much more frequently. Adding small heatsinks will always help affected areas, but from what I can tell, setting up some active fans make the biggest difference. Whenever you are using cooling fans, make sure to keep them well maintained. Check for dust accumulation and make sure the area around them is free of debris (use a filter/grill if possible to protect the fan blades). Broken fan blades can add a lot of noise to your system and if the cooling fan fails, your board will end up overheating again.

If your board is constantly overheating, or you are using a RAMPS board with thousands of hours of printing on it, you will want to replace it entirely. RAMPS boards are normally extremely cheap at under \$10, but higher-end boards may be closer to \$150. These higher-end boards will experience less overheating when wired correctly.

Luckily boards on newer machines shouldn't have this issue, though it is still worth checking to see if the stock fan on your board has stopped working.

Summary of Fixes and Precautions

- Confirm you are using the correct Volt/Amp for your Heater/Board.
- Check for visual damage (burnt out cords/connectors).
- Test out a new heater if you do not see any damage.
- Solder wires or rewire any section that is not connected or burnt out, always using the proper gauge.
- Replace any burnt out connector to the board.
- Actively cool your board.
- Replace board if overused or burnt out.

Hotend Not Reading Correct Temperature

This failure refers to your hotend reading a different temperature than what it actually is due to issues with your thermistor, and can lead to bigger problems if not properly addressed. This is noticeable when filament feeds at different temperatures than it is supposed to, or if you are receiving a message of “Error Min Temp”.

The instructions in this chapter are similar to the “Build Plate Not Reading Correct Temperature” chapter and is addressed in the “Electrical Safety” chapter as well. At the end of the day, if your hotend is not reading the correct temperature it is a thermistor problem.

Error Min Temp (or Error – Stopped Temp Sensor)

If your thermistor is not connected properly your printer, it will stop working and read “Error Min Temp”. If it doesn’t give this error, you could have an issue with not having thermal runaway setup which is why you should read the “Electrical Safety” chapter. Having this error read out is most common on fragile, non-cartridge thermistors, but can happen on just about any setup.

Instead of “Error Min Temp”, some printers may read either a negative temperature or 0°.

If your printer is reading one of these errors you will need a new thermistor the majority of the time. You can order replacement thermistors from your hotend manufacturer. I personally use E3D hotends, and you can easily order replacement cartridge thermistors from their website, Filastruder.com, or Matterhackers.com for US orders.

If your thermistor has no physical damage that you can see, you may be able to fix this without buying a new one. Inspect your thermistor cords for any visual damage or disconnections.

If there is a frayed wire or a section of a wire that has no continuity, you can cut out the burned area and solder the two wires together wire, or rewire entirely. If your thermistor is still intact, replacing the wiring will likely fix your issue. You will then need to confirm that the extruder can reach its farthest point from the board, since you just made the wire shorter than it was.

If your newly soldered wire cannot reach the board at the extruder’s farthest point, you will experience layer shifts on large builds, or possibly rip the thermistor off the hotend.

It is generally recommended to rewire entirely instead of cutting and soldering.

Not reading proper temperature

This issue is extremely difficult to diagnose if you are not looking for it and when the differential is not extreme. When drastically different from the actual heat you can have a major problem on your hand if the hotend does not stop heating and your printer does not have thermal runaway.

If your thermistor is not reading correctly, or if it is hovering an inch next to your hotend, your printer will think that the hotend is not able to reach its set temperature. Your printer will continue to heat the hotend, or just shut off if you have thermal runaway setup. I have personally seen melted metal from a heater block that never recognized it was actually getting hot, since the thermistor was not connected to it.

You can imagine if the thermistor is dislodged or not properly connected to the hotend, it would read a temperature much lower than the hotend actually is.

Other less severe symptoms may include consistent nozzle clogs or over/under extrusion. If you are only printing PLA you may not ever notice the temperature is reading incorrectly, due to the fact that most PLA has a wide extrusion temperature range.

At the end of the day, you want to make sure you have a good thermistor and that it is wired properly to your board.

Summary of Fixes and Precautions

- Keep a spare thermistor for your hotend on hand if possible. I have gone through dozens of thermistors over the years.
- Make sure the thermistor is connected to the hotend. If hovering next to it your hotend may not stop heating and can cause serious problems.
- If the thermistor is noticeably damaged, replace it.
- Check for breaks or frays in your wire then rewire as needed.
- Make sure that if you do rewire you give enough slack to ensure that the extruder can move to its farthest point.
- Replace and rewire thermistor from the hotend all the way to the board.
- Flash firmware to factory settings.
- If still not reading the proper temperature, you may need to replace your board.

Layer Bulges

The issue of layer bulges have only happened to me when using Ender 3 and Ender 3 clones, though I assume it could be possible on many machine types, especially those with only one leadscrew.

I have a recently played around with a lot of trial and error, but there seems to be one main culprit: a leadscrew that isn't perfectly 90 degrees.

This is due to either a misplacement of the z-stepper motor or misplacement of the area the leadscrew is threaded into the mount, or both. This causes the leadscrew to be slightly angled from a perfect 90 degrees. There are a few solutions to this problem:

Remove Z-hop

This is likely the smartest solution for most inexpensive printers. Once I removed a Z-hop, those layer bulges went away entirely. I have recently learned that inexpensive machines should not have a Z-hop and should be reserved for well-built machines or Delta printers.



In the above photo, I have two prints that have exactly the same settings on an Ender 3 V2, but the one on the left has a Z-hop and the one on the right does not.

This squished layer happens because it is more difficult for your stepper motor to raise in the Z-direction than to lower. When you have a Z-hop of 0.2mm, your printer may only raise 0.19mm, then drop back down 0.2mm, causing a squished layer.

Luckily Cura and other slicers allow for a setting called “Avoid printed parts when travelling” which should nearly remove the concerns of nozzle drag or knocking a part off. If you have that checked (it is in the “Travel” section on Cura), then there shouldn’t be much of an issue.

This is definitely the easiest fix for this layer bulge problem and what may work the best for you.

Check leadscrew to see if it is bent

Take your Z-leadscrew and lay it on a flat table. Roll it back and forth and see if the rod is straight. Any bends will be noticeable when rolling. If this is bent or curved, then unfortunately there isn't much you can do other than order a new leadscrew. Just make sure to purchase the same size and pitch as your current leadscrew, then upon replacement you should hopefully notice a big difference.

Clean and lubricate your Z-rod

As mentioned with mandatory maintenance, you should keep all of your rods lubricated. I would recommend removing the leadscrew, then cleaning it entirely in isopropyl alcohol to remove all the gunk.

Then grab some white lithium grease, or your preferred lubricant, and cover the entire threaded rod. This will allow for easier movement and less likelihood this issue will occur.

Check to see if leadscrew falls straight into Z-stepper coupler

Loosen your stepper motor coupler and lift the threaded rod out of it. Raise your carriage to at least the midpoint, then thread the leadscrew down and see where it falls into the coupler. If your leadscrew doesn't go straight into the coupler, this can cause your leadscrew to not be perpendicular to the build plate and lead to layer bulges.

It is fairly common on Ender 3's for your leadscrew to line up with the back of your coupler, as if your stepper motor needs to be pushed back a few mm. This can be fixed by adding a spacer to the stepper motor mount, so that your stepper is set back slightly. There are new mounts you can print for your stepper on Thingiverse, or you can add your own spacer.

If your leadscrew hits to the right or left of your coupler, this may require a new stepper motor mount that is adjustable. With these mounts you can adjust your stepper left or right to line up properly with the leadscrew.

If that doesn't work, or you would like to try another method, check the next step.

Purchase a Spider Coupler

Spider couplers allow for compensation if your leadscrew doesn't line up perfectly. You can get a set of two for about \$15 and are well worth it if you are having this issue.

Bend the Z-Carriage to be 90 degrees

One of the main problems is that the mount that holds your Z-lead screw nut and attaches it to your frame is slightly bent and not a proper 90 degree angle. I have watched videos where individuals actually just hammer and bend the carriage back to the proper angle, though I have not personally tried this method. A video titled "Creality Ender-3 Z-Axis Alignment Correction" by Ronald Walters on YouTube goes over this exact method. This should result in fixing the problem, though it may be hard to visually see if you are at a proper 90 degrees without tools that will help.

You can also just reprint the entire body. I tried this by printing the design by hunterius_prime on Thingiverse, though this unfortunately did not solve my issues.

Add a second leadscrew

This costs a bit of money, but adding a second leadscrew can compensate for any issues with your other leadscrew. Having two leadscrews is a good upgrade to have regardless, so it's something to think about. That said, the main issue is due to the manufacturer not having the leadscrew lined up perfectly through the mount and the stepper motor.

This means that the only real solution is to fix this alignment. Adding a spider coupler or printing a new mount for your stepper isn't a perfect solution, though it may work. This can be very difficult because it is hard to even see if the leadscrew isn't a perfect 90 degree angle with your naked eye, meaning it is hard to tell how far or in which direction you need to change these things.

If you are able to tell how your leadscrew isn't angled properly, then you can fairly easily design a new mount for your z-stepper motor, making the leadscrew line up perfectly with your mount.

Add an anti-backlash nut

If your leadscrew isn't the issue, you can try adding an anti-backlash nut. One individual I spoke to was able to completely fix his problems by doing this. So if your leadscrew is properly lined up and not bent, then it may be worth buying this inexpensive item to prevent your carriage from dropping when it shouldn't.

All of the above said, the easiest solution is just to remove your Z-hop. Removing the Z-hop and adding "Avoid Printed Parts when Traveling" will drastically improve your print quality if you are having bulge issues.

Summary of Fixes and Precautions

- Remove your Z-hop. This is the simplest solution and should help a lot. Make sure you turn on “Avoid printed parts when traveling” in your slicer to prevent parts being knocked over.
- If your leadscrew is bent or curved, you will need to purchase a new one.
- Clean and lubricate your leadscrew.
- Compensate for a leadscrew that doesn't line up perfectly with your coupler by either adding a spacer to your stepper mount, printing an adjustable stepper mount, or purchasing a spider coupler.
- Bend the metal carriage that holds onto your nut that attaches the leadscrew to your X frame so that it is a 90 degree angle.
- Add a second leadscrew which may be a little costly, but will help compensate and overall make your printer better.
- If your leadscrew is nice and straight and lined up and you are still having this issue, an anti-backlash nut may help.

Layer Shifts

Layer shifts refer to when the print looks fine other than the fact that one or multiple layers are shifted in the X or Y direction. Layer shifts can be something as simple as a loose wire or can be as difficult as recognizing that your stepper motor pulley is not functioning properly.

This can result in a print that has one, or multiple layer shifts. Much of this chapter is similar to previous editions, just with some tweaks to make the process a bit smoother with modern machines.

Single Layer Shift

This issue is normally a bit easier to diagnose and fix than a print with multiple layer shifts.



Obstruction during print.

The most common cause of a single layer shift is that there was an obstruction during the printing process. This can be from tangled or too tightly wound filament, or from a cord that is the way of an axis from moving properly. You will see this occur more frequently on larger prints than small ones.

It is another reason I suggest only buying highly made filaments from reputable manufacturers. I have actually had brand new filaments that had a tangle in the middle of the spool, causing a layer shift mid print. I have heard people say this is impossible – but it has definitely happened to me.

You will want to make sure that your printer has a clear path before starting a print, and that all cords and wires are not in a position to obstruct after moving throughout the entirety of your print area. Confirm your filament is tight on its spool and that you maintain it in a way that will not allow it to tangle. If you have a spool that is becoming unwound during the print, get some Teflon or PTFE tubing and mount it to your frame as tight as possible to help.

Zip tie all cords in a fashion that get them out of the way of the toolpath. Anything in the way of the extruder or build plate that is stronger than your stepper motor will cause a skip, and then result in at least one layer shift.

Endstops in wrong spot, or frame not set up properly

You will run into issues with a single layer shift if you are printing something large and your full print area is not set up properly. If you are using a slicing

program where your machine settings are not to the proper dimensions of your printer, the machine will think that it can print further than it actually can. The stepper will skip when your extruder or bed hits its max build area, and the print will continue, assuming that it went the entire tool path.

You can check to make sure everything is set up correctly by homing your machine. When you home the printer and it goes to the very corner of your build plate, you likely have everything set up properly (or the center for printers with a home at center).

When you home the machine, if there is accessible print area in front of or to the side of the build plate (depending on your homing setup), you are not going to be able to use the entirety of your build area.

This can easily be fixed if your X or Y endstop is just in the wrong position by slightly adjusting their location. If these endstops are fixed on your machine, the frame itself may not be set properly. Adjust any t-nuts that may be holding your bed in the position it is in and slide until in the proper homing position. Retighten and make sure nothing can rattle. You need to make sure that your printer homes in the correct spot in order to take advantage of the entire build area. You would never know there is an issue on small prints until you decide to go with a large G-code.

This can also arise if you recently swapped your hotend setup. Your printer is designed to reach its max points with the stock hotend, and if your new hotend or mount pushes the nozzle in any direction from where the stock one was, then you will no longer be able to reach the max build area of your printer. You will need to either relocate your end stops, or adjust your build volume in your slicer to accommodate. Otherwise you will think you can print a larger model than you actually can, resulting in a single or multiple layer shifts.

Errors in G-code or model

You will want to check out the “Model Errors” chapter for a further description, but essentially files can be corrupted or exported improperly. While in your slicing program check the model layer by layer to see if there are any holes or missing walls. Also be sure to check for actual errors on software such as Cura or the old version Netfabb.

Your G-code can actually be corrupted as well. This is not common but I have had prints that just would not print properly no matter what I did. This normally happens if you transfer a file before it has completely saved.

This is a lot harder to diagnose, but if you have a part that has caused a layer shift in the exact same spot after reprinting, then it is worth your time to

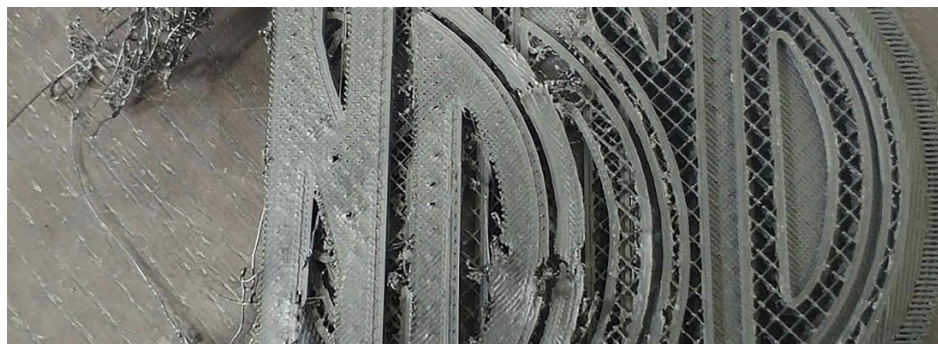
reslice and reupload to your machine. It would be very unlikely to experience a layer shift in the same position if it were just caused by an obstruction.

SD cards can also become corrupt. Try formatting your SD card or use a new one if continually having a problem.

Too thick of layer heights – turn combing off/use lines for infill

This should not be an issue when working with standard nozzle sizes and layer heights, but can become an issue when going over 0.4mm layer heights. When I tested out the SuperVolcano with a 1.4mm nozzle and 1mm layer heights, my nozzle would drag over the previously laid infill when traveling. Turning off combing allowed for the hotend to Z-hop after every movement and avoid this infill. I also switched from triangular infill to lines, in order to avoid the same problem. When I did not do this – I got a layer shift from the nozzle hitting the infill and skipping the stepper motor.

Multiple Layer Shifts



Belts too loose (or too tight)

Belt harnesses on many machines are built in a way that will cause loosening over frequent printing. Some inexpensive machine harnesses only hold the belt tight via a zip-tie, though luckily this has not been as common as of late.

A loose belt will cause slippage and excess play. This is a very common problem when you are using a heavy bed that moves back and forth frequently (you will see frequent layer shifts in the Y-axis).

You will want to make sure your X and Y carriage have belts that are very tight. You can actually over tighten them, but from my experience, a loose belt is far more common than one that is overly tight. If your belt is too tight it may cause binding and ghosting (as covered in that chapter).



It is smart to print an adjustable belt tensioner for your carriages. This will allow for easy tightening when things get loose over time – otherwise you will likely have to disassemble. Luckily printers like the Ender 3 V2 come with belt tensioners stock, but if yours doesn't have one, I definitely suggest adding one. I go over this further in the “Z-Axis Wobble” chapter. After adding one of these you actually can over-tighten, so be careful.

There is no specific measurement to judge if your belts are properly tight, I usually just say you don't want any droop and want the belt to be springy to the touch. If the belt feels as though it's stretching and has no real give –

you've gone too tight.

Bed corners tightened to their max

This is actually far more common than you would think, especially if you are not using a bed leveler. When attempting to get a level build plate you may run into a time that you end up over-tightening one or multiple corners. You do not want these corners at their maximum spring tightness because over time it can actually warp the metal plate. These warps will make your problems even worse.

You will notice an issue when you try to move the Y carriage (on Cartesian setups) with the printer off and stepper motors disabled. When one or multiple corners are over tightened, the bed will be difficult to move.

I suggest starting fresh by loosening all of the corners until they have equal minor tension on the springs. If you now notice a big difference in how easily your bed moves, then this is likely the culprit. Get the Z-rods even by checking the distance differences for the nozzle to the bed in the X direction. Just hold one rod in place while you twist the other, leveling the X-carriage (as explained further in the "Unlevelled Bed" chapter, assuming your printer has dual Z leadscrews).

Only then should you adjust the corners for a level build plate. If you have a very warped metal plate you can experience certain corners that will just not get level no matter what you do. In this instance you will actually need a new metal plate, though a bed leveler may help. This is why you do not want to leave any corner over-tightened for long periods of time. You will be slowly putting pressure on an item that may get bent over time. Using a thicker 1/4" glass build plate helps to make this issue less common. Fulament actually has a build plate I recently tested that advertises just how flat it is, and it really does help to never worry about this issue. Just make sure the four corners are equally tensioned and you shouldn't have to worry about one corner being further from the nozzle than another. The flatter your build plate is, the easier it will be to avoid this issue.

This is just one of the many reasons an auto bed levelling device (such as the EZABL by TH3D or the BL Touch) may be worth your time. I personally do not use them on many of my machines, but they can be great for a deformed build plate or just for someone new to this 3D printing hobby.

Dry rods or broken bearings

Most printers have self-lubricating bearings for their carriages, but even those can get dry over frequent machine use. This is for printers moving over smooth rods, such as the Prusa. If you are having difficulty manually moving

the X or Y axis when the machine is not printing and stepper motors are disabled, check to see if the rods are extremely dry or that the bearings are not broken. A broken bearing is easily replaced and a dry rod can be fixed with some white lithium grease. Also check to make sure that the rods themselves do not have scarring from the bearing wearing down. If the rod is heavily scratched, then that may need to be replaced as well.

Just rub a minor amount of white lithium grease to the rods (both threaded and smooth) and then move the carriages around so that it spreads. If you notice a drastic increase in smooth movement of your axes, then you may have fixed your issues of multiple layer shifts. I have actually noticed some big difference and reduction in layer shifts after doing this in the past on machines with thousands of hours of run time.

Reapplying lithium grease and checking the resistance on the rods/bearings is good practice regardless of experiencing layer shifts, since it can help to allow for consistent clean prints.

If you have a printer that has rollers moving over aluminum extrusions, such as the popular Ender 3, then you will want to make sure your rollers aren't clamped too tight to the frame. This really hasn't been too much of an issue for me, since being too loose is far more common. But if your rollers are clamped so tight that it reduces movement, then you will want to loosen it slightly.

These rollers are held on by a nut on one of the rollers. This nut does not tighten when turned clockwise and loosen when turned counter clockwise. Instead one side of the nut will result in a loose carriage, one side in a tight carriage. So if you do a full 360 degree turn of this nut, you will be right back at your starting point. If you notice that the rollers are held on too tight, give the nut a 90 degree turn and then test again. You don't want the rollers free spinning, so make sure it is tight enough to not have free play but still be able to roll smoothly.

Bent rods

Rods, especially thin threaded 5mm Z-rods, can become bent over time. This bent rod can cause one or multiple layer shifts as the carriage or bed travel over these bends.

If you notice any rods that are bent, replace them immediately.

Acceleration or speeds too high

Your motor's torque at a given speed must be greater than the force needed to accelerate or decelerate the carriage at a given acceleration rate and

maximum speed. If you require a higher torque than the motor can supply at that given speed or acceleration, the layers will shift via the motor skipping.

Most printers do not have the acceleration settings on the LCD screen anymore, but some do. That said – it is much easier now to just edit your acceleration settings in the slicer itself. Most slicers, including Cura, have this ability to edit your accelerations in the “Speed” section. You likely do not want this number over 1000 when working with Cartesian machines. CoreXY and Delta machines can get much higher depending on the build quality. Manually reduce this number and see if it helps. I actually have some of my machine set to 500m/s/s acceleration even though it can likely handle higher, since I want consistently good quality, non-ghosted prints and am willing to wait the extra time required. CoreXY machines can handle a higher acceleration than Cartesian, meaning I have my CoreXY set to between 1000 and 1500. Delta machines can handle up to 3,000 without much of an issue. You can obviously increase from these numbers if you know your printer can handle it, such as many of the new Voron builds.

You can also find these numbers via a G-Code command through Repetier Host, Octoprint, or another terminal you are using. You will be fed out the current stock numbers if you type in the command “M503”. That said, I still believe it is easiest to just tweak and set your acceleration numbers using your slicer.

If you are ever having extreme difficulty with a printer and want to flash back to factory settings, but you do not have the version of Marlin for your printer, you can also type “M502”, which is a factory reset. Remember you will lose any settings you may have changed (such as E-steps), but this has saved me in the past when I was unable to determine why my motors were performing improperly.

After setting anything new for your firmware via G-code commands, you will have to type “M500” to store the settings. If you are using something like the Ender 3, you will need an SD card in your machine, since you will not be able to save your settings without one. This means your settings will reset when you turn your machine off and on again without having an SD card.

Driver current is too low or too high

Having the current going to your stepper drivers too low can cause insufficient motor torque and result in layer shifts/stepper skips. A driver current that is too high may overheat the stepper and cause a thermal shutdown. This shutdown will result in that stepper no longer working until it is cool once again. The same is true if the driver on your board is too hot.

Make sure that you have a fan on your board, a heat sink on your stepper motor, and a small heatsink for your stepper drivers. You can then check the current via the method described in the “Extruder Stepper Skipping” chapter, but with the X or Y motor in question. This should not be an issue with factory made machines, but will be more common on the inexpensive DIY ones, though I have had to change the stepper driver on one machine from the manufacturer that was experiencing this overheating issue. That said – I have read articles from individuals that have had this issue on stock Creality machines, so worth checking the VREF if you think your steppers may be underpowered.

You first want to have your machine turned off and disconnect the stepper motor cable from the board. You then need to look up both the stepper motor you are using, as well as the stepper driver, or just search online for VREF values on your particular printer.

Current limits are determined in the motor and driver data sheet. You will not want to run higher than either the driver continuous current limit, or the motor current rating limit, so it is often good to have a driver that has a higher continuous current rating. I suggest going off of the continuous current limit of the motor.

Once you know what current limit you want, you then need to find out the calculation for your stepper driver to determine a VREF. You can go to the current limiting section of your stepper driver data sheet in order to figure out the VREF you will want. Current limit will equal $VREF \times 2.5$ for standard A4988, and $VREF \times 2$ for DRV8825. There is then another option by TMC where the calculation is slightly different, where $VREF = (\text{Motor current} \times 2.5) / 1.77$. To find out your ideal VREF, there is a handy calculator that someone made online which you can search for by typing “Stepper Driver VREF Calculator”, where you just enter the rated current of your motor and it will read out the ideal VREF numbers for each driver type.

That said – it might be difficult for you to figure out exactly what your machine needs, so it might be smart to search online for your specific printer and its recommended VREF or voltage. This helps a lot since many people

have already figured out proper settings for popular machines. Going off of TH3D's website, an Ender 3 V2 should have a VREF in Volts of these numbers for the X and Y axis to prevent layer shifts:

X: 1.1 – 1.2V

Y: 1.0 – 1.1V

I understand this is very confusing for someone who is new to this, just as it was for me. But essentially the VREF is the power that is being sent to your motors and it can be tweaked depending on your motor and stepper driver setup. So if your max current limit on your motor is 1AMP, and you are using the standard A4988 driver, you will have a VREF of roughly 0.4. This is because $0.4 \times 2.5 = 1$. This 0.4 would be your target number that your multi meter will read out. If your VREF is lower than 0.4, you will not be having enough power sent to the motors, meaning they can result in layer shifts and motor skips. Going higher than 0.4 can result in an overheated motor.

When dealing with the Ender 3V2, you would want your multimeter to read out a number in-between the ones provided above for a proper VREF.

To test this out, you will actually need to plug back in your power and turn the machine on, while you still have access to the board. Be careful now that everything is on. Make sure the driver you are testing has the stepper motor unplugged from the board. You would then grab your multimeter. No real way to do this without a multimeter.



Set your multimeter to 20V DC, and touch your black negative lead to a ground pin on the stepper driver (titled GND). If you are unaware which the ground pin is, you can also touch the black negative lead to the negative section of your power supply. On my Ender 3 V2, I just touch the metal on the screw terminal where the black cable is going from my board to my power supply. Just make sure to only touch a ground section with your black

negative lead.

You can then clip the positive lead of your meter to the metal shaft on the screwdriver to help read everything out while you change it. If you do not have a clip to connect the lead to the screwdriver, you will need to test, tweak, and test again. You then touch the positive lead (or the screwdriver if you have it clipped) to the potentiometer on the stepper driver. This is a very tiny screw like object on the driver. You will then see a voltage number on your multimeter. Remember you need your printer powered on and your stepper in question disconnected, so make sure you only touch a ground and your potentiometer. This is your VREF, which you want to make sure equals your calculation above or whatever the proper rating is for your machine.

You can turn the screwdriver clockwise to increase the voltage, and counterclockwise to decrease (which is actually the opposite on some drivers, so just make sure you are testing after each small turn). A 1/8 turn of the potentiometer will make a drastic difference in your VREF – so make sure to not turn too fast. Turn a very, very small amount, retest, and then continue until your multimeter is reading the correct voltage.

Once you get the proper readout for your printer, plug everything back in and run another test print. If your VREF was originally set too high, your stepper motors should run much cooler now. If you were getting skips and layer shifts and your VREF was set too low, that problem should hopefully be fixed now.

Stepper still skips

If your stepper still skips, you may have a malfunctioning motor or have wiring issues. Check the connectivity for each wire going from your stepper to your board. If there is a break somewhere you will have to replace it or cut it and rewire. Sometimes one wire will be connected, and then become disconnected upon certain movements, since it is frayed or damaged. This will cause the stepper motor to not turn properly, and then turn properly again once the wire has continuity again. Make sure every wire from your stepper to your board had proper continuity.

The stepper or driver itself may be malfunctioning, so try switching the connectivity to a different axis and see if the motor still skips when moving that axis. If it does, replace the stepper to fix your layer shifting and skipping issues. If it ends up your stepper driver is malfunctioning, you can easily replace that. If you have a printer such as the Ender 3 where the stepper is integrated, you unfortunately would need a whole new board. Make sure you confirm it is the stepper driver though before going this route, and not just the stepper or a continuity issues.

It also may be worth re-flashing your firmware or going back to factory settings if this issue came out of nowhere. Flashing firmware has become much easier on machines such as the Ender 3 V2. For those machines you would just search on Creality for the most recent firmware for your particular board and printer, download and unzip the .BIN file, then transfer that .BIN file to your SD card. Turn your printer off, put in the SD card, and then turn the printer on. Your printer will now flash the most current firmware.

Remember that anything you changed will now go back to factory settings, such as the E-steps. This method is only possible with newer printers, such as the Ender 3 V2, but not possible with the older version 1. For something like the Version 1, you will need to bootload, which there are tutorials online for. I recommend watching TH3D's videos on the topic if you need to bootload.

Cheap or worn-out pulleys

Pulleys for your motors need to have sharp defined teeth with the proper spacing in order to work properly moving your belt the correct amount of steps. They also need to be tight on the stepper motor shaft, since any slippage on this will cause a layer shift.

If you buy a poorly made pulley, or notice that yours have had the grooves worn down, you will want to upgrade and purchase a new set. Go for products made from aluminum or stainless steel made products, since small items such as this are not that much more expensive, and can make a huge difference. If you do purchase a new pulley, make sure it has the correct amount of teeth. If you replace a pulley with the improper amount of teeth, your parts will all be to very incorrect dimensions or cause further layer shifts.

Confirm pulleys are tight on the stepper motor shaft

Your stepper motor pulley is held onto the stepper motor shaft via a grub screw (also called a set screw). If this set screw is not tightly holding the stepper motor pulley, then the stepper motor itself can turn without turning the pulley, or just slip slightly from time to time. This means that your printer will think it's moving, but the belt isn't turning, or at least not the proper amount.

Make sure that your grub screw is tightly holding the pulley onto the stepper motor shaft and that there isn't any free play at all. If there is still free play even when you tighten, replace the grub screw or replace the pulley itself until it can be held on as tight as possible.

Bed too heavy

This normally is not an issue if you have your drivers putting out the proper current and everything is lubed, but if you have an abnormally heavy bed you may experience shifts in the Y-Axis (on Cartesian machines - this should not happen at all on a CoreXY machine.). If you are trying out an experimental bed with a lot of wiring and extras on top of a ¼" thick glass bed, this axis may have difficulty moving without experiencing some layer shifts.

For this issue, you will either need to replace your build plate to be a lighter one or check the VREF on your Y stepper motor to confirm it is sending enough power. If you confirmed your VREF is correct, other than replacing to a lighter build plate, you would need to swap the Y stepper motor be a more powerful one.

This really hasn't been an issue for me in a very long time, so make sure you confirm that the build plate can move freely when stepper motors are disengaged before thinking this is the culprit.

Running into the print

A less common problem with layer shifting can be when the printer head runs into the layer it just printed. This can cause a skip in the motor and will have the print continue where it left off. You are more likely to get a print knocked off the build plate in this occurrence, but sometimes it can cause a layer shift if the part is stuck to the build plate extremely well.

This is most common when you are over extruding at very low or very high layer heights with a very strong bed adhesion.

As explained in the “Parts Being Knocked Over” chapter, you will want to add a Z-hop to that of your layer height, that your hotend is assembled tight and not oozing, and you will want to make sure you have your printer head avoid printed parts in your slicer settings. As mentioned earlier in this chapter, if you are printing at very large layer heights, you may need to tweak your infill pattern and turn off combing.

Z-hops aren’t recommended for inexpensive machines, so first try using the “Avoid printed parts when traveling” in your slicer. If you still get your nozzle hitting your print, then you can try adding a Z-hop.

Z-wobble

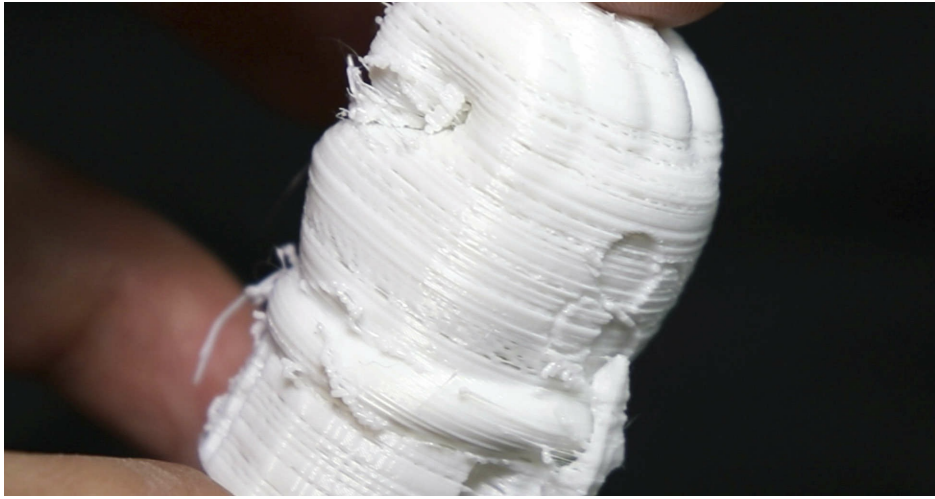


Some may consider Z-wobble in the class of layer shifts, but I have it as its own category. If you are experiencing a wobbly looking print in the Z-Axis (never ending extremely small layer shifts), please refer to the “Z-Axis Wobble” chapter

Summary of Fixes and Precautions

- Clear the printing path of your carriages from any obstructions. – Zip tie all wires and loose cords, and maintain a clean printing area.
- Confirm end stops are in the correct spots and that the frame is built correctly so that when you home the nozzle it is in the furthest part of the corner that it can be.
- Check for errors in model or reslice if G-Code is corrupted.
- Increase Z-hop, turn off combing, use lines for infill to prevent nozzle from hitting the print on large layer heights. This should only be relevant for very large layer heights.
- Make sure belts are tight enough (but not too tight).
- Do not over tighten bed corners.
- Make sure the rods are not dry or that any bearings are broken.
- Replace any bent rods.
- Reduce your acceleration and/or speed.
- Increase the current going to your stepper drivers by checking the VREF.
- Check to see if your stepper or drivers are malfunctioning or overheating.
- Make sure your pulleys are attached tight to your stepper motor shaft and cannot spin freely.
- Replace or upgrade your pulleys.

Missing Layers and Holes in Prints



This is an issue that occurred for me randomly and I had a lot of trouble diagnosing it. I actually have an old video on this titled: “Diagnosing and Fixing an Unknown 3D Printing Failure” on my YouTube Channel. This particular problem can also be known as “temporary under extrusion”, though it can also be as minimal as seeing some holes in your print.

Replace poorly made extruder

If you were to watch my video listed above, you would find out that replacing my extruder is what finally fixed this issue for me. This is not an ideal solution, but I was using a home-made Greg's Wade extruder, which likely had tolerance issues and may have worn down over hundreds of hours of printing.

If you are using an extruder that you personally made, or one that is made with lack luster tolerances, I would suggest changing it. As mentioned elsewhere I have standardized to Bondtech or E3D, and it is well worth it in my opinion.

At the minimum you should replace any home-made extruder with one made by a reputable manufacturer if this problem is occurring (which can be found for under \$20 for non-geared). You can always reprint your home-made extruder parts after fixing this problem.

Check for extruder skipping

One of the main reasons this will occur is if your extruder motor is skipping. I have a chapter on this topic, so you will need to read that if your extruder is skipping and making a “clicking” noise.

Check that your extruder gear is attached to your stepper motor

The hobbled gear that is attached directly to your extruder stepper motor shaft is normally only held on by a small set screw. If this set screw isn't pinching tight onto the extruder stepper shaft, then the stepper itself will turn while the gear is not. If it has a light hold on your stepper, it may turn sometimes and not others, leading to gaps in your layers. If it doesn't have a grip at all, then your printer will look like it is printing, but no material is being fed.

Disassemble your extruder and make sure the set screw is tightly holding on this gear onto your stepper. If it spins freely or has any freeplay at all, you will surely have under extrusion or just missing layers. This is a pretty common reason temporary or complete under extrusion.

Holes at the end of every layer

Coasting is a great addition to slicers which allow for the last section of an extrusion path to be a travel path instead. This means it takes the oozed material and uses it to finish your layer, as to reduce blobs.



Coasting should only be used when working with a Bowden printer. If you have coasting turned on for a direct extrusion printer, you will inevitably have small holes when each layer finishes, as shown in the photo above. You can actually see these holes in your slicer when you turn on “layer mode”. These holes are filled in by Bowden printers oozing, but you won’t experience this extreme oozing when using a direct extruder. This means you will just have holes after each layer finishes.

You can also have coasting set too high, so reduce the numbers if experiencing holes on a Bowden printer. Perhaps your Bowden is set up perfectly with the exact retraction settings you want, so just try turning coasting off if you are still having these holes at the end of every layer and see if the problem no longer occurs.

This problem is very common when you upgrade your extruder to be direct and you don’t change your slicer settings to accommodate. Retraction settings are much lower on direct extruders and you will want to make sure you turn coasting off.

Check Teflon/PTFE Tube on non all-metal hotends

I always recommend all-metal hotends, but the vast majority of inexpensive printers will not come with one. This means that your PTFE or Teflon tube will be pushed all the way to the heater break of your hotend. This is why you can't print very high temperatures, since it will melt this tube.

If your PTFE tube isn't pushed all the way into your hotend, it can cause minor clogs or issues that can lead to gaps in your layers.

Confirm pulleys are tight on the stepper motor shaft

Your stepper motor pulley is held onto the stepper motor shaft via a grub screw (also called a set screw). If this set screw is not tightly holding the stepper motor pulley, then the stepper motor itself can turn without turning the pulley, or just slip slightly from time to time. This means that your printer will think it's moving, but the belt isn't turning, or at least not the proper amount.

Make sure that your grub screw is tightly holding the pulley onto the stepper motor shaft and that there isn't any free play at all. If there is still free play even when you tighten, replace the grub screw or replace the pulley itself until it can be held on as tight as possible.

Extruder idler tension

The idler on your extruder is what is creating the tension on your filament. Some basic extruders may not have one, but all extruders will have some form of spring that puts tension so that your filament is held tightly between the threaded bolt and the bearing (or both threaded bolts for dual drive extruders).

Confirm that the filament is held tight so that no slipping can occur. If you can easily push or pull filament through the extruder when the extruder has full tension, then you likely need to increase the tension. It shouldn't be very easy to push or pull filament through an extruder that you are not manually keeping open. One reason for missing layers is that the filament is not held tightly enough.

Turn up your extrusion temperature

This can happen on a print when your hotend isn't hot enough for the particular material you are printing. The PLA I normally use has a temperature range of 205-220 degrees Celsius, and I almost always print at the higher range, right around 215 degrees. I experienced this issue once on my main printer with upgraded parts since I was printing at 205 degrees. Right when I bumped this back up to 215, the issue went away.

Printing at too low of a temperature for your material can cause too little of filament to extrude. And this can happen at random times during your print rather than throughout, since the hotend doesn't have enough time to heat the material when going at its top speeds. This would result in temporary under extrusion and missing layers.

Slow your print down

Just as with running too low of a temperature, you can be feeding your filament too fast for either your extruder or your hotend.

When working with a stock non-g geared extruder, along with a stock hotend, I wouldn't run my printer any faster than 45mm/s, no matter the material. With my upgrades I can easily print 60mm/s or higher, but whenever I run prints on a stock lower end machine, I decrease this to 45mm/s or less. If I am experiencing any issues, I go closer to 35mm/s.

This is a common solution to many problems in this book. Slowing your printer down can not only help you to diagnose particular problems, your printer may actually require you to slow down. Don't always believe manufacturers advertised printing speeds.

Make sure your fan isn't dropping your printing temperature

If your active cooling fan is blowing onto the heaterblock and nozzle, rather than right below the nozzle, you can experience your extrusion temperature dropping in the middle of your print. Since we said that having your printing temperature too low can lead to this problem, your active cooling fan dropping the temperature can be the culprit as well.

Always use a silicone sock if possible on your heaterblock, since they will help to prevent any fluctuation. I use them on every one of my E3D hotend prints. If possible, re-print your active cooling fan to one that blows downward and wraps around the nozzle. Search on thingiverse and elsewhere for a file if you are unable to design one yourself.

Be sure to read the “Hotend Can't Reach or Maintain a Temperature” chapter if this is occurring for you, since there are a few solutions including running a PID autotune.

Filament Diameter Problems

You should always confirm that your filament diameter in your slicer settings match what you have on your spool. This should always be set to either 1.75mm or 2.85mm, since those are the only two standards used in 3D printing.

That said, your particular manufacturer of filament may not have very tight tolerances. The tolerances of materials can be anywhere from .01mm to 0.1mm.

You can use calipers to confirm the average diameter of your material, but in all honesty, I would just suggest going with a more reputable manufacturer with tighter tolerances. Most reputable manufacturers produce filament with tolerances between .03mm and .05mm these days which is adequate for most prints.

Potential under extrusion

For this particular issue, I would assume under-extrusion isn't the main problem, but it could be adding to your problems. Since having a missing layer is temporary under extrusion, the rest of your print should be extruding properly.

Check your E-steps, just as explained in the "Over and Under Extrusion" chapter. This likely wouldn't fix the problem by itself, but the symptoms may be reduced.

Re-slice your part

If this is happening across multiple prints, then obviously re-slicing won't fix your problem. But if you are only using the same G-Code, and the problems keep occurring at the same layer, it is worth your time to re-slice and export new G-Code.

Before exporting, go into the layer mode of your slicer and analyze. I have actually had in the past where you can see the missing layer right in the slicer. This was due to a model error, and after fixing it, the slicer then showed that layer being printed.

Check your rods/rails, bearings/rollers, and lubricate

Be sure to look up and down your printer to see if there are any issues, and perform a full maintenance check as explained in the “Mandatory Maintenance for 3D Printers” chapter.

One thing that chapter details is for you to lubricate your rods. This obviously isn’t needed on rail systems, but all rods should be lubricated for bearings to move easily.

With your stepper motors disabled, move your hotend and build plate around its axis and check for any rough spots or where there is more friction than others. If there are rough spots, then you will need to lubricate, check your frame for any bends or misalignments, confirm your build plate corners are tightened too tight, and make sure your bearings/rollers aren’t broken.

You may need to replace your bearings and/or re-align your frame to make sure everything moves freely.

Extruder motor overheating

While this has never personally happened to me, I can only assume that your extruder motor, or stepper driver, overheating can lead to temporary under extrusion.

When stepper motors or stepper drivers overheat, they will not turn or work until cooled down to a working temperature. If this happens on your extruder stepper motor, then it won't turn properly, under extrude, and then kick back on. If you swapped from a large extruder motor to a pancake stepper, you may need to reduce your VREF.

Your extruder stepper is working harder than any other axis, since it is under very high loads when forcing filament through the nozzle. If the stepper is running hot, you want to make sure that you have a heatsink attached to the motor, and a small one attached to your driver on your board.

Refer to the “Stepper Motors Overheating or Malfunctioning” chapter if this is happening to you for a full fix, since it involves checking your VREF.

Summary of Fixes and Precautions

- Replace a cheap or poorly made extruder.
- Check for extruder motor skips, and refer to that chapter if occurring.
- Make sure the gear attached to your stepper motor shaft is held on tight and that the set screw is doing its job.
- Turn off or lower coasting if you are experiencing holes at the end of every layer.
- Ensure your PTFE tube is pushed all the way to the heatbreak when using a non all-metal hotend.
- Turn up your extrusion temperature to closer to the top of the recommended range.
- Slow your print down, especially when using a stock non-geared extruder.
- Confirm that the printer isn't dropping temperature mid print due to the active cooling fan.
- Confirm you are using the correct filament diameter, and that you are using from a reputable manufacturer tight tolerances.
- Check your E-steps as to not exasperate the problem.
- If occurring on one model, make sure to re-slice and examine the layers mode to see if the slicer is showing the problem.
- Do a physical mechanical check of your printer and perform all important mandatory maintenance.
- Make sure your extruder stepper or stepper driver isn't overheating. If so, refer to the "Stepper Motors Overheating or Malfunctioning" chapter.

Model Errors

I am personally not a designer, nor am I extremely familiar with all the designing software out there, so I am not the best for giving advice when it comes to combining parts and properly exporting them in your preferred program. That said, there are some common issues that come into play with models exported for 3D printing. This chapter is very similar in its explanations as previous editions of this book.

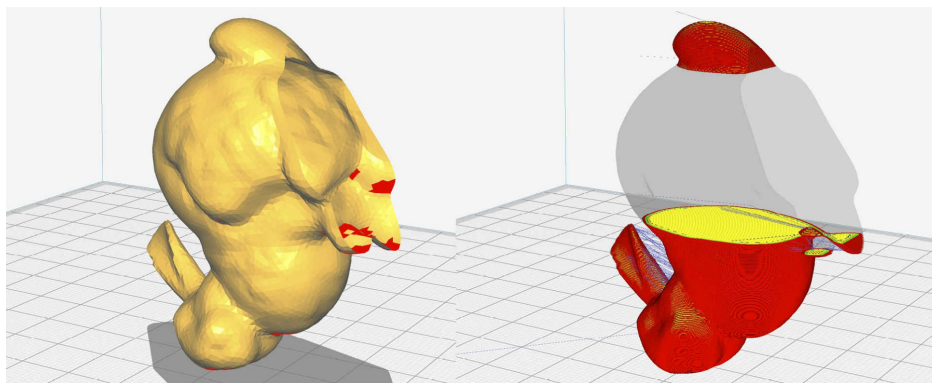
Holes, one sided walls, and other model errors

Other than the problems described in the next sections, there can actually just be errors within your model itself. This is extremely common if you used SketchUp to do your designing, which I do not think is very popular anymore. I do not have the exact reasoning for this, but it seems to happen with .STL's exported from SketchUp more so than any other program.

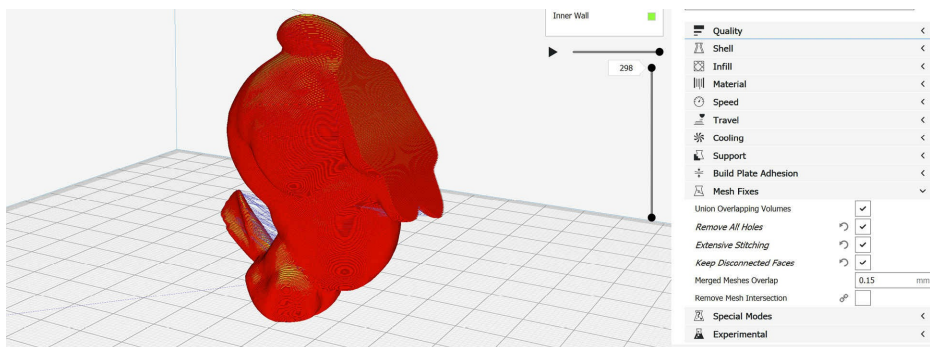
Someone who takes all of the proper steps to design a model for 3D printing, including combining and exporting properly, should never experience a model with errors. But there are some free and paid programs out there that can help you diagnose and fix model errors.

I personally use the free version of Netfabb Basic. This actually isn't available any longer (at least without some hunting on GitHub), but there is also the ability to fix models via Cura. Netfabb Basic is actually about 3 or 4 years old now, but for some reason I still like using it.

Below is an example of a model missing a face and being too thin to print since it is not solid.



When you open up your settings you are given “Mesh Fixes” as a settings option. You can play around here and go back and forth from layers view to see if your problem was corrected.



If you are able to download a previous version of Netfabb, this is also a very easy process. I was able to download this old version via some hunting online and finding a version of Netfabb Basic on GitHub. When you bring an .STL into Netfabb, you will see a red exclamation mark telling when there is a model error. You would then click on the “Repair” button up on the top toolbar

Choose “Automatic Repair”, execute the default repair, and then click “Apply Repair”. This fixes the vast majority of model errors so that it can be 3D printed.

If you still see the exclamation mark after attempting this repair, or you can’t figure out how to download Netfabb Basic, you can also try out Microsoft’s free online model repair tool at tools3d.azurewebsites.net. I haven’t used Microsoft’s tool in a while but it should still work, since I just checked as of editing this book. There are likely other free methods to fix models, such as using Meshmixer, but I personally do not use those. Your designing software itself may have the ability to repair as well.

If you are still experiencing issues, you may need to redesign or re-export the part.

Parts that are not actually 3 dimensional

A common practice for those who do not have much designing for 3D printing experience will be to take a 2D object and fold it around itself.

If you were to take a 2 dimensional rectangle and then fold it around itself so that it is a cylinder with an open top and bottom, the object will look 3 dimensional to the human eye, but will not be recognized as such by the 3D printer or slicing software.

This is because the object you are looking at has walls that are infinitely thin – they do not have any actual depth to them.

You will need to close off these objects and make them solid, or do some other designing practices that allow them to become actual 3 dimensional objects with solid walls.

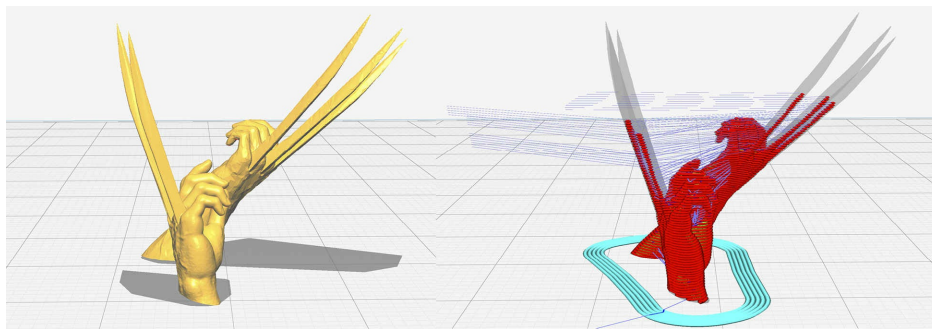
Wall thickness

This wall thickness issue may not be considered a technical error in the model, but essentially result in the same problems. You will always need to consider the thickness of the nozzle diameter you are using when designing the thinnest part of your print.

An FDM 3D printer can only print walls that are at least as thick as the nozzle diameter (or rather, the line width). I personally almost always print with the same line width as the nozzle diameter, though I have been experimenting with increasing it by 10% lately. This issue is also explained in the “Limitations with 3D Printing” chapter, so you can skip this if you already read that.

I normally suggest walls be at least 2-3x as thick as the nozzle diameter/line width, but the absolute minimum requirement is the diameter/line width itself. So, if you were using a 0.4mm nozzle (with 0.4mm line width), but some of the walls on your print are 0.3mm thick, those walls will not be recognized, and therefore will not print.

Along with increased detail, a smaller nozzle will allow you to print walls that were not visible on nozzles with larger diameters. You can always see if your walls are recognized by your printer by checking the “layer mode” in your slicing program.



In the example above, the tips of Old Man Logan’s claws (designed by Exequiel Devoto on MyMiniFactory) are thinner than 0.4mm – the diameter of the nozzle I was using. Increasing the size of this part will allow you to print more, since the diameter of the blades will increase. If you are unable to print larger, you will need to use a smaller diameter nozzle or switch to resin printing.

If something is thinner than the thinnest diameter nozzle available, you have a part that cannot currently be FDM 3D printed. Well, that isn’t exactly true anymore - Cura also now has the ability to “Print Thin Walls” in the “Walls” section. This will actually make it so your printer will recognize and print

these walls. That said – there is no real way that they would be to the proper dimensions, though it is definitely an improvement over older slicers.

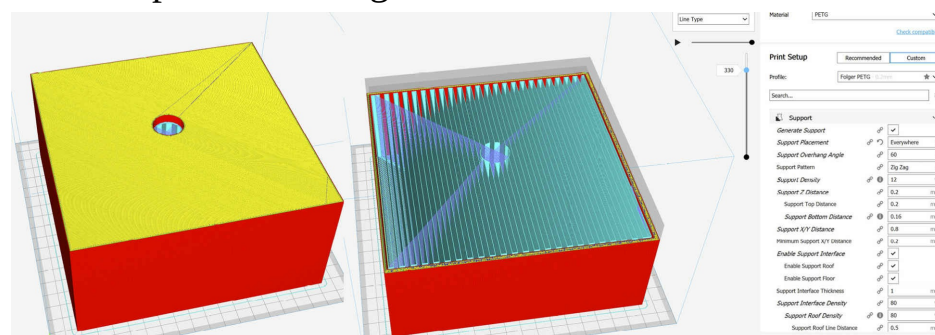
I have further information on the pros and cons with printing in small diameter nozzles in a video titled, “3D Printing with Extremely Fine Nozzles”. I often suggest going with resin printing if your goal is very thin and small parts, since not only are they far easier to print with this method, they will come out much cleaner and of higher quality.

Unprintable models

A model can be considered unprintable for many reasons, one of which being it requires support material that is impossible to remove. This is also explored in the “Limitations with 3D Printing” chapter in this book.

If you were to design a hollow cube with a single small hole giving you access to the inside, you will likely never be able to print this model, unless you were using dissolvable support material with something like an IDEX printer.

This is because a cube of any decent size will require either infill (not hollow), or removable support material. If your hollow cube does not give you access to the inside, you will not be able to get that parent support material out (unless the support material is dissolvable). The other option would be to attempt to bridge the large gap, but no printer or material will be able to bridge 100mm+, and the shell walls of the center hole will be floating in mid-air. This is why this hollow cube may be possible if small, but just about impossible if large.



You can see this a bit better with the above example. It would be just about impossible to remove that support through the hole, but you wouldn't be able to bridge this large gap without support material. If you were using dissolvable support material, you could soak the part and then slowly let the goopy material fall out of the hole. This may be one of the rare cases dissolvable supports are required.

While this basic model may have no actual errors, you will not be able to print it successfully.

You can see my video titled “Cura Tricks for 3D Printing” where I am able to print one of these unprintable models, but it was quite difficult and may not work for all models.

Summary of Fixes and Precautions

- Try to avoid designing on Sketchup.
- Always combine parts and export them using the proper methods of the program you are using. Do not cut corners.
- Use Cura, Netfabb, Meshmixer, or a similar program to diagnose and fix printing failures due to poor models.
- Make sure the walls of your print are at least as thick as the diameter of the nozzle/line width you are using, or at least have “Print Thin Walls” checked in Cura.
- Inspect your print in your slicing programs “layer mode” to see what toolpath the printer will actually follow.
- Always take note if you have the ability to remove the support material that is being laid down.

Not Finding Home and Inverted Prints

When you start a print your extruder and bed will go to its “home” position. It does this by moving the carriages until hitting their respective endstops. If there are any obstructions in the way of this process, or if there are any malfunctioning endstops, you can run into some serious structural issues.

The other way this can happen is via your firmware being set up improperly. If your printer is going to the incorrect location for home, you may also be experiencing inverted prints. This should not be an issue on a stock, pre-built printer.

Remember to refer to the “Diagram of a 3D Printer” near the beginning of this book in order to know what and where endstops are.

This chapter is similar to very similar previous editions of this book.

Clear axes of any obstructions

Each axis and machine setup will come with different obstructions that can prevent your printer from finding home. When this happens, your stepper motor for that axis will continue to turn and skip even though it cannot move any further. If this is on the X or Y axis, you may not experience any long term damage to your printer, but when on the Z axis you can have a nozzle that continues to dig into your build plate, resulting in issues to your frame, nozzle, and possibly even cracking your bed.

The most common obstructions for both the X and Y axis are going to be wires. This is why it's important to have all of your wires cleaned up and out of the way of any axis movement. If you have a wire blocking your build plate from hitting the endstop, then it will make a lot of noise attempting to turn the stepper motor. Really anything can become an obstruction, so make sure your axes can all move and engage the endstops.

One reason your X endstop won't engage would be if you recently swapped out your hotend and/or extruder. Your stock setup may have been designed to trigger the endstop, but your new carriage may not. You will need to add something, or redesign the carriage, so that the endstop gets engaged with your new setup.

The Z axis really does differ on every machine and setup, but if you are using a mechanical screw to check your Z-height, there is a possibility it has been dislodged and not engaging the endstop. Confirm that the screw lines up properly with its endstop and that it is able to engage it to the point that the endstop clicks. This isn't as much of a problem today as the past, since auto bed levelers have become more common.

Clear any obstructions and confirm all endstops can be engaged when homing before moving on to the next solution.

Malfunctioning endstop

A malfunctioning endstop is one that does not send a signal to the controller (ie. light up) when engaged (pushed down until it clicks). This could be due to a broken endstop, or due to the wiring being disconnected.

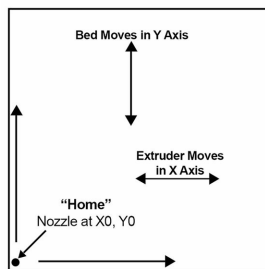
First confirm that all wires are plugged into the end stop and board, and that none have been pulled out from their connectors. Then check for any easy to see visual issues such as frayed or burnt out wires. If you do not see any, pull out your multimeter and check each wire for continuity. If one wire does not read out any noise or continuity, attempt to find where the break is, cut and solder, or replace the wire entirely.

If all wires show connectivity, your endstop itself may need to be replaced. Luckily, these are very cheap and I recommend having a spare on hand to test and fix this issue fast. If you do not have a spare on hand, you can test this easily by connecting to a different endstop. If it lights up when engaged, then you will need to replace the endstop in question. If it does not light up, then the endstop is not the issue, and more likely has to do with your wiring.

Homing to wrong side, or endstop on wrong side

These next steps should only really be an issue on a machine you are building yourself. If this is occurring on a machine that was sent to you from the manufacturer, it was likely their fault, and worth contacting the manufacturer.

3D printers can have very different understanding of where “home” is depending on the manufacturer. For some - home is when the X carriage is all the way to the left and the bed is all the way back, others it can be the exact opposite. The Ender 3 has “home” on the front left of your build plate like the image shows. The Ender 5 has “home” be the back right. CoreXY printers will have the bed homing by either moving it up to the nozzle, or down to a specific set point. Though this can vary, the standard for the majority of Cartesian (non CoreXY or Delta) machines is having the nozzle home to the front left side of your build plate:



If you have your endstop on the incorrect side for where your machine is homing, it will clearly never be able to find it. You will hear a lot of noise as the extruder or build plate continues to try and move and cause your stepper motor to skip.

If you are unaware of where “home” should be on your printer, it is a good bet that it is where your printer moves to when starting a new print. If you are sure that home should be on one side, yet your printer always moves to the other, you may have your settings incorrect in either the firmware or when slicing your G-Code. You will definitely be able to tell if this is the case if your prints are coming out inverted.

If your prints are not coming out inverted, it is likely easiest for you to just physically change the location of your endstop. If you would rather go the firmware route, open up Marlin when re-flashing your machine, and go to the Configuration.h tab. Scroll down until you see “ENDSTOP SETTINGS:” followed by each axes home direction being defined by either 1 or -1. If the axis that is homing on the incorrect side for you, and it says 1, change it to -1. If the axis’s that is homing on the incorrect side for you says -1, change it to

1.

```
// ENDSTOP SETTINGS:  
// Sets direction of endstops when homing; 1=MAX, -1=MIN  
#define X_HOME_DIR -1  
#define Y_HOME_DIR -1  
#define Z_HOME_DIR -1
```

Inverted prints

If your prints are coming out inverted, it is normally when one of your axes is inverted via firmware, slicing program, or your motor is wired/setup improperly. You would really only need to change one of these regardless of which was causing you this issue. If you setup your stepper motor on the wrong side, so that it is turning your belt the incorrect direction, it will likely be easier to just invert this axis in your firmware rather than rebuilding that part of your machine.

To do this open up marlin to flash your machine. While in Configuration.h, scroll down to where it says “#define INVERT_X_DIR” followed by either true or false, and the other axis’s below it. Whichever axis is inverting for you, switch the language (turn “false” to “true”, or vice versa).

```
#define INVERT_X_DIR false    // for Mendel set to false, for Orca set to true
#define INVERT_Y_DIR false    // for Mendel set to true, for Orca set to false
#define INVERT_Z_DIR true     // for Mendel set to false, for Orca set to true
```

If you are currently homing properly, but your axis is inverted, you will want to change the language on both sections of the Configuration.h tab mentioned above. This will correct the inverted axis but still allow home to be in the same location, meaning you do not need to relocate any endstops. If you only change the invert, but not the endstop settings, you will be homing in the incorrect spot.

When finished, plug in your machine, choose the correct board and port, and upload.

Another option is to literally invert the wires going to your stepper motor in question. If your wires are in the reverse order, the stepper will turn in the opposite direction. This will of course change where “home” is, so you should only do this method if you are homing in the wrong spot AND have inverted prints.

Summary of Fixes and Precautions

- Confirm that there are no obstructions from any axis hitting its respective endstop.
- Confirm that the endstop lights up when engaged.
- If the endstop does not light up, check that all wires are showing continuity. Replace any that are not.
- If all wires show continuity, replace the endstop in question.
- If homing to opposite side of your machine but parts are not inverted, you can either physically move your endstop to the other side or you can change the proper language in Marlin.
- If homing to opposite side of your machine but parts are inverted, change the proper language in Marlin
- If parts are inverted but you are homing to the correct side of your machine, change both the language in both sections of Marlin mentioned above.

Nozzle Clogs

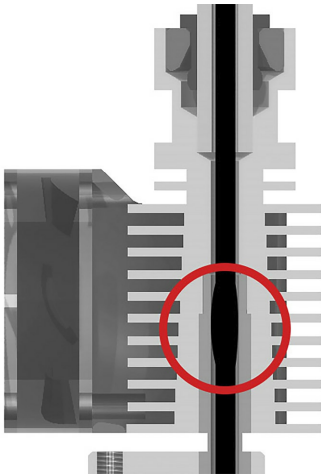
Nozzle clogs are just what they sound like and are a fairly consistent problem with FDM machines. They can take a very long time to fix and can normally not be diagnosed until the print has been going for a while. Most of the time these nozzle clogs will actually be clogs in the barrel, but we will still refer to this issue as a nozzle clog. We can think of these as more “filament clogs” where the filament is clogged somewhere in the hotend.

If the nozzle clog is frequently happening, even after doing these fixes, you will need to review the “Settings Issues”, “Hotend Can’t Reach or Maintain Temperature”, and “Hotend Not Reading Correct Temperature” chapters.

One way to help prevent nozzle clogs is to use an all-metal hotend. Many inexpensive machines do not come with an all-metal hotend, and have PTFE tube go all the way to the heat break. This can cause unwanted issues and further nozzle clogs, especially when consistently printing near the max those hotends can handle. An upgrade to an all-metal hotend is not only needed for printing at higher temperatures, but recommended to just avoid nozzle clogs.

IMPORTANT NOTE: Anytime you switch filament you will ALWAYS want to push the material down around 1 cm before removing. Some minor heat creep can occur when the printer is cooling down after a print and is only noticeable when you attempt to remove it. This practice can save you from an untold amount of barrel clogs. I never change filament without pushing some out before removing it.

Heat Creep



Heat creep refers to when heat creeps up the filament and causes unwanted expansion. If filament within the barrel gets too hot, it causes the plastic inside the barrel to heat beyond its glass transition temperature, and will often create a little bubble. This bubble will cause a ghost print where the printer continues to try and print, grinding filament or skipping your stepper, and result in filament that you cannot remove.

Please keep in mind that this used to be a common issue for me, but over the past few years it has only happened a handful of times. If you take all of the proper precautions outlined in this book, it should not become too much of a problem.

Heat creep is far more common on materials with a low glass transition temperature, since it allows the material to deform at a lower temperature. Be sure to refer to the “Material Science” chapter in this edition for a further explanation on this, and how PolyMaker has been able to work out a PLA that doesn’t run into this issue.

If the heat creep is minor, you may be able to use a small Allen wrench to push the filament through while the hotend is hot until it makes it past this bubble, but you do not want to put too much pressure on your machine or else you will cause extra problems for yourself.

When the heat creep is more excessive, you will want to disassemble your hotend. If your filament hasn’t already grinded to the point where it has snapped, you can break it off now. Disassemble as far as you can until you are left with the section that will not allow you to move the filament. This is normally in the barrel, but can be in the heaterblock itself.

If in the barrel, you will want to remove all plastic parts that may be connected (fan, printed adaptors for your machine, PTFE tubing), and any wiring that you can. There shouldn’t be any direct wiring to the barrel, but

you will need to remove any PTFE tubing from inside if using 1.75mm filament. If you remove this tubing, or have an all-metal hotend, you will then want to get a lighter or a propane fueled torch. Be sure to not use a high temperature torch due to the possibility of it melting your barrel. You can use a standard Bic lighter if in a pinch.

Before proceeding I must warn that this is dangerous and should only be done at your caution. You will want to be in a very well-ventilated area and wear a mask for your protection. Do not attempt this procedure if you do not feel comfortable handling the equipment. Use gloves and take proper precaution.

Hold the barrel with a wrench or pliers that have rubber grips up to the flame (while wearing the proper gloves). Doing this with a wrench that does not have metal grips will cause the handles to get far too hot.

If possible, pinch the filament in-between another set of pliers with your other hand. You can then carefully pull until the barrel gets hot enough that the filament will melt and the material will be easily pulled or pushed through. Clean out the barrel from remaining debris if you see any by pushing a small Allen wrench through.

If the clog is in the unlikely section of your heaterblock and not your barrel, you will want to do this same process, only you must make sure all wiring is out of the way or removed. This type of clog is rarer because the hotend can normally heat itself hot enough to push filament through without the need of a torch.

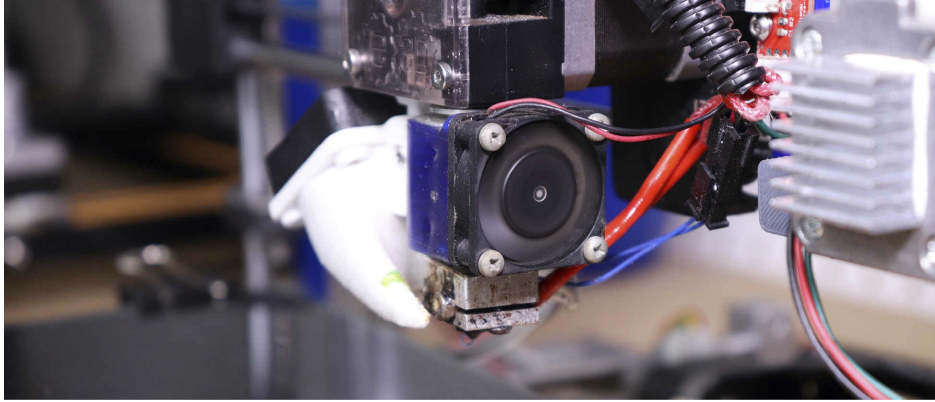
Be sure to allow the barrel to cool all the way down to room temperature before reassembling. Once reassembled you can now move forward with future prints, but be sure to read on to prevent this from happening in the future.

How to prevent heat creep:

Since heat creep means that the temperature from the hotend is creeping up the material, there are a few prevention tips that can be tried.

Confirm your barrel cooling fan is on and not obstructed:

All hotends should have a fan that blows on the barrel of your hotend. I wire my barrel cooling fan directly to the power output on my board so that it is always on, something you may want to consider (and most manufacturers have built-in). If not wired directly, it should turn on when your hotend gets over a certain temperature – around 50 degrees.



If this fan is not blowing or is obstructed by debris when your hotend is heating, you will need to fix it in order to prevent heat creep. The same is true if the fan itself has broken blades. Some of these fans do not have shields over them, and I have accidentally bumped one or two in the past with an Allen wrench. This caused a blade to break off and not have as good of airflow – meaning a replacement fan was required.

First turn off and unplug your machine. Make sure your fan is of the proper voltage for your machine (12v/24v) and that all the blades are intact. If you have an extra fan of the proper voltage available, connect that fan instead and turn back on your machine. If it works, you can skip the next section since the issue was a burnt out fan.

If the above solution does not work, unplug and turn your machine off once again. Inspect the wiring of your fan. You may easily see that a wire has come loose or been disconnected. If you do not see a problem, you will have to use a multimeter tool to check the continuity of the wiring from your board to the fan. If one of the wires does not show continuity, you will either need to re-solder the unconnected section, or rewire entirely.

This is simple on many machines, but can be quite a headache on others. Having the proper connectors and ability to solder will make this process exponentially easier. Be sure you see connectivity on the wire all the way from your board to the fan, there are no exposed wires, and turn your machine back on.

If the fan still does not turn on, there may be a problem with the section of the board that that fan is wired to. With the machine unplugged and off, wire the barrel fan directly to the power input section on your power supply (red to + and black to -). Turn your machine back on, and if you are using a new fan and new wiring, everything should be working again.

Links to where you can purchase fans are at my website 3DPrintGeneral.com

Make sure your PTFE tube is pushed all the way

on hotends that have one

If you aren't using an all-metal hotend, which will be true on almost every stock inexpensive machine, there will be PTFE tubing that goes into your hotend all the way to the heat break. If this tube isn't pushed down far enough, then you will have a high likelihood of expansion of your filament where you don't want it.

Make sure that your PTFE tube is not damaged and that it is pushed all the way down.

Make sure you are not printing too slow or too hot:

This will involve you checking the settings on your slicing program. If you run your hotend too hot, too slow, or a combination of the two, it will allow heat to move up the filament easier. Try lowering the heat a bit or speeding the print up to see if this helps with the problem. Normally lowering the print speed will help to diagnose and fix issues, but when combined with a very hot nozzle it can create a bit of heat creep.

This is especially true with hotends that aren't all-metal. That PTFE tube can only reliably print on hotends at 240 degrees or lower, and going hotter than that will likely deform the tube. This deformation will not allow filament to feed properly.

Reference the "Settings Issues" chapter of this book for further help with tweaking your slicer settings for a more accurate print.

Reduce retraction settings

If your retraction settings are too high for your extruder and hotend, it could cause something similar to heat creep, where the deformed bottom of your filament gets trapped when being retracted back up. This has really only happened to me when using the Hemera extruder/hotend, since that setup requires very low retraction settings, though I can imagine it can happen with any setup if set too high.

Since there is such a small gap between the extruder and the heat break on Hemera setups, having a high retraction distance can lead to somewhat of a clog. For a Hemera and similar style setups, I have my retraction distance set to 1mm or less, quite a big difference from Bowden and even other direct extruder setups. When I had this set to 3mm, I would run into failed prints due to clogging.

Swap or dry materials

Poorly made materials and those that have absorbed moisture will lead to more issues with clogs. If your spool is old, you may want to dry it out - I have methods listed in the “Materials and their Settings” chapter.

You could also just have a poorly made spool by a non-reputable manufacturer. Buy your materials from those with lots of reviews – some of the ones I like are listed in the “Resources” chapter.

Final attempts to fix:

If you have done everything above and are still experiencing heat creep, you may want to invest in a new barrel or an entire new hotend. I have personally used E3D hotends for a while, and while they do have an occasional heat creep, it is not a consistent issue. Your barrel may have been degraded over time causing the diameter of the hole to be inconsistent. Or the company you may have purchased it from may not have the tolerances and quality that other products guarantee. I have seen other makers cut an E3D hotend in half and compare it to off-brand knockoffs, and you can easily tell the tolerances are much more precise on the name brand E3D version. Sometimes you get what you pay for.

Finally, your thermistor may be reading the incorrect temperature. Though very uncommon, your hotend may think it is 230°C, when it is actually running at 250°C. To fix this issue, please refer to the “Hotend not Reading Correct Temperature” chapter of this book.

Clog in the Actual Nozzle (clog but you are able to remove the filament)

A clog in the nozzle and not your barrel is normally caused by one of three issues: settings issues with the diameter of your nozzle, extruding at too low of a temperature, or improper purging of the previous filament. You will want to treat a clogged nozzle in the same way you would with a clogged barrel that I go into further detail above: Remove the nozzle and carefully apply a torch in a well-ventilated area until you can push the clogged filament through. Remember to always remove and replace nozzles with the hotend set to 240°C due to the expansion of metal.

Too fast for the diameter of nozzle you are using:

It's easy to understand that the larger diameter of nozzle you are using, the easier it is to print without any clogs. E3D makes nozzles from as large as 0.8mm in diameter (up to 1.4mm in their SuperVolcano nozzles) all the way down to 0.15mm. I have personally used every diameter of nozzle they offer and can personally attest to the increase in nozzle clogs the lower you go. It took me roughly two dozen failed prints on their 0.25mm nozzle before I zeroed in the proper speed and retraction settings for this nozzle diameter without any clogs or issues.

This meant that I torched out the nozzle quite a few times before I got everything right, each time having to wait hours into the print before the issue showed itself. I repeated this process with the 0.15mm nozzle, and with each type of material. Once zeroed in, you should only use the same manufacturer, since each company you purchase filament from may require slightly tweaked settings, and these minor differences in settings become more apparent the smaller in nozzle diameter you go.

There is no one step solution for this - you will need to manually tweak your settings until you achieve a print without any clogging (though upgrading your extruder may help). Refer to the "Settings Issues" chapter of this book for further detail on how you may be able to fix this yourself. You can always move up in nozzle diameter if you can't seem to get things to print properly. I have found that a 0.4mm diameter nozzle achieves very accurate prints with very minimal issues of clogs, and a 0.6mm nozzle is extremely easy to use and fast to print, but not the greatest when it comes to X/Y accuracy of text and other details.

If you want to print really fast with large nozzle diameters – you will need a hotend that allows for this, such as the Volcano setup by E3D. This is because the standard hotend does not allow enough time for the material to heat up when extruding so fast – something the Volcano compensates for.

If you really want high detail, then it would be worth looking into resin printing, and you can refer to the new chapter in this book for my introduction to it.

Filament Grinding

When this failure gets extreme, it can lead to you needing to read the “Stripped Filament” chapter. This is likely due to having too much moisture in the material.

If you are using flexible material, you may see the filament curled up before it grinds. Make sure you slow the print down if this happens, and confirm you have an extruder with enough torque for flexible material. Also confirm that there is a straight, clear path through the hotend. Bowden machines cannot print many flexible materials, even when using a strong geared extruder. The longer the distance the tubing runs between your extruder and hotend, the more difficult it will be to print with flexible materials. This is just one reason the Hemera, and extruders like it, have become so popular. With close to zero gap between the extruder and the heat break, you can print soft materials at the same speed you would print PLA.

How to prevent filament grinding:

Loose or tight idler tensioner, blockage, wrong hot end temperature, too fast of retraction and print speeds are all common causes - yet easy to correct. The one that is the most annoying is from having too much moisture in your filament. Make sure you also read the “Stripped Filament” chapter in this book.

If you were to speed up your machine 10x what it normally prints at, you can imagine how your extruder gear would be turning far faster than material can be pushed out the nozzle. This would cause the filament to grind until the point of snapping (if not just causing your extruder motor to skip).

Before clicking re-print, try slicing your part with both your print speed and retraction speeds slower. Watch the print and see if there are any sections where the extruder is moving too fast, and attempt to fix in your slicing program or by changing the feed rate on your LCD. The general rule of thumb is to limit the maximum print speed to 100x the nozzle diameter with a stock extruder, printing at no greater layer heights than 75% of the nozzle diameter. The print speed maximum is a starting point, and with a good geared extruder on a well-built frame you can get closer to 175x the nozzle diameter, if not faster.

Next, adjust your idler tension. Start by loosening the idler and then feed filament through and tighten until you no longer experience slipping, but not so tight you experience grinding or stepper motor skipping. Filaments vary in diameter, so although the idler will absorb some variations, some material will require fine adjustments. I have my idler extremely tight on most of my

extruders now since the grip and torque on that are good enough where I do not experience extruder motor skips, but you will have to find a happy medium with a lower powered extruder.

You should also check the temperature you are running your hotend at and make sure it is not running too cold. If you try to print ABS at 200°C, you might be able to get some filament to extrude at first, but it will eventually get stuck. If slowing your printer down didn't help, try increasing the hotend temperature a bit (which is the opposite advice given for heat creep).

Summary of Fixes and Precautions

- Push filament down before pulling it out.
- Don't keep material heated for long periods of time without printing.
- Make sure the barrel cooling fan is working and that all the blades are intact.
- Confirm wiring from the board to the barrel cooling fan is connected properly.
- Clean your barrel cooling fan and remove all dust.
- Push PTFE tube all the way in for a hotend that isn't all-metal.
- Do not print higher than 240 degrees on a hotend that isn't all-metal.
- Check your settings that you are using the proper extrusion temperature for the material you are printing.
- In a well-ventilated area, torch metal parts when required.
- Make sure your thermistor is reading the correct temperature.
- Adjust the idler tension.
- Tweak the print speed.
- Confirm you understand what is said in the "Settings Issues", "Hotend Can't Reach or Maintain Temperature", and "Hotend Not Reading Correct Temperature" chapters.
- Read the "Stripped Filament" chapter if experiencing this issue frequently.
- It is smart to have replacement heater blocks, heaters, and thermistors so you do not need to wait for delivery to fix your machine when required.
- Only use parts from reputable manufacturers. Tolerances may not be right on knockoffs.
- Make sure you aren't printing at too low of a temperature for your particular material.

Over/Under Extrusion

Over and under extrusion is normally caused when your stepper motor is not turning the proper amount of steps, and results in either too much, or too little plastic being extruded. This will lead to ugly and/or brittle parts.

Keep in mind that a minor amount of under or over extrusion can be unnoticeable to the human eye, so it is smart to check this intermittently regardless if you are experiencing issues (as mentioned in the “Mandatory Maintenance” chapter). Unnoticed under or over extrusion can cause issue with parts that need to fit together or parts used for mechanical purposes.

Note: For most of your issues, you will want to skip down to the Check and Fix your E-steps section of this chapter.

Step 1: confirm you have the proper filament diameter set

This sounds silly – but it has happened to me. Without thinking, I have had my filament diameter set to 2.85mm on Cura, but I was actually using 1.75mm. This will 100% result in a print that looks far under extruded. This can be more common than you think, because for some reason Cura still has the filament diameter set to 2.85mm when you set up a custom printer.

The opposite is true if you are using 2.85mm filament but have your slicer set to 1.75mm – your part will look massively over extruded. This likely isn't as common though, since 2.85mm diameter extruders are not nearly as popular as they once were.

Check this before doing any further solution explained, since you will just be wasting your time. This is an easy fix – just make sure you have this set correctly in your slicing software.

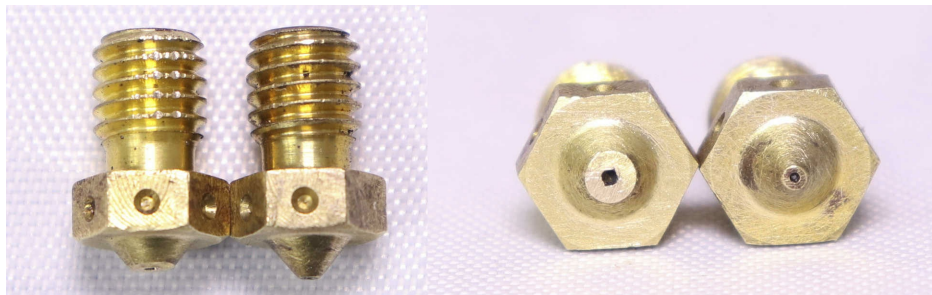
Over Extrusion

Drastically over extruded parts will be fairly easy to tell either to the naked eye, or easy to tell because parts are not fitting together. If you factored in your machine tolerances (which I go over in the “Parts Not Mating Together” chapter) and parts are still not mating together, you likely are experiencing a bit of over extrusion.

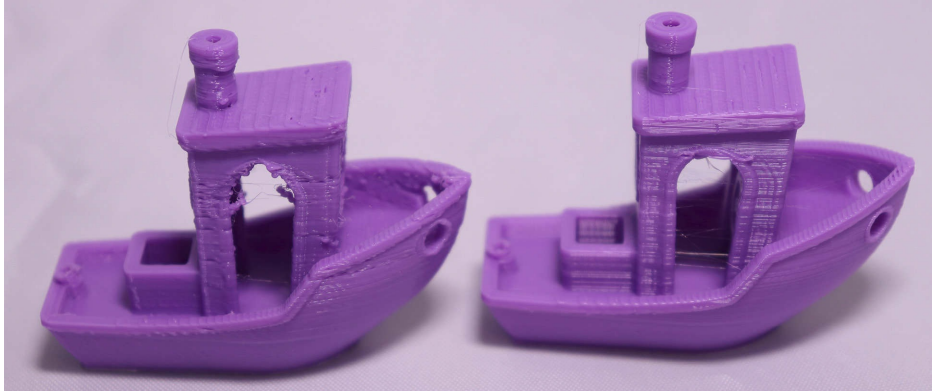
Check to see if nozzle is degraded

Nozzles can become degraded over time from material constantly running through them, as well as from times you started a print too close to the build plate. Some materials, such as the carbon fiber reinforced blends, require a hardened steel nozzle to prevent rapid degradation. I have standardized to using E3D Nozzle X on my printers, though they just announced a new ObXidian nozzle, which is supposed to be even better.

When your nozzle degrades, the diameter will get slightly larger than it was when factory shipped to you. This can lead to some parts of your print looking as though there was over extrusion (or may just look generally ugly). You won't really be able to tell if you have this issue, unless your nozzle looks like the one on the left below. The nozzle on the right has been used, but it clearly has not been as degraded as the one on the left. Worn out nozzles will lead to ugly prints far before they get as bad as the one pictured below:



Normally a nozzle won't be this noticeably degraded, so you will want to keep track of how much printing you are doing on a machine. If you have gone 6 months of daily printing on a brass nozzle, it is definitely time for a replacement. It is smart to always keep 1-2 spares on hand to test if this solves your issues of over extrusion.



Above the two parts have the exact same G-code, but the one on the right is with a new nozzle. The part on the left was printed with a hardened steel nozzle, so even these wear out after hundreds of hours of printing with abrasive materials. While the print on the right is not perfect, you can see just how much a nozzle with the proper tolerances will help to achieve clean prints.

You should also check your E-steps, as explained further in this chapter, though a brass nozzle with a few hundred hours of printing should always be replaced. It is also well worth the investment to get a hardened steel nozzle so you do not have to worry about this issue for a long time. I personally have not used one due to the price, but you can even go with a ruby tipped nozzle to never have to worry about this issue again (they cost roughly \$100 though).

Under Extrusion

Unless extreme, under extrusion is much more difficult to diagnose with the naked eye. This is because parts will bridge gaps just fine, the surface quality will look good, and parts will mate together with ease. The biggest issue comes with the integrity of your parts. They will delaminate easier and will break under far less pressure than the part should be able to handle.

Check tension on extruder idler

Make sure there is enough tension on your filament by the idler that pinches the filament to your hobbled bolt/gear. When this idler is too loose, you may experience less filament feeding through the hotend than there should be. Too tight and you will experience further skipped steps in your extruder motor, or even stripped filament.

Confirm your hobbled bolt or gear is tight to stepper motor shaft

Take your extruder apart and see if your hobbled bolt or gear that is attached to your stepper motor shaft does not have any free play. Just like I explained in the “Missing Layers and Holes in Prints” chapter, this gear is held onto your shaft via a grub screw (also called a set screw). If this grub screw is loose, then your extruder stepper may turn but not actually extrude any filament. This can lead to no material coming out at all, or just lead to under extrusion due to slippage between the gear and the stepper motor shaft.

Confirm that there is no free play at all on your extruder gear that is held onto your stepper motor shaft.

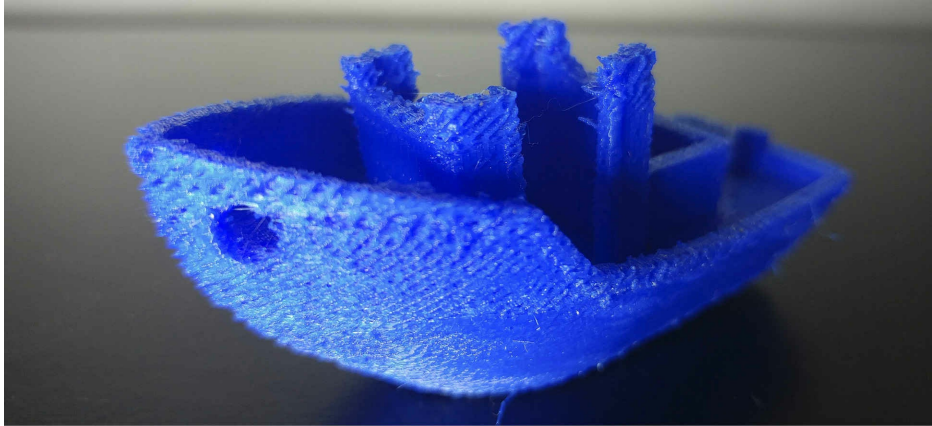
Are you experiencing extruder motor skips?

If you notice that your extruder is making a “clicking” noise, then the stepper is skipping. Please refer to the “Extruder Stepper Skipping” chapter to fix this issue – since an extruder that skips will cause under extrusion.

Not enough torque to extruder for nozzle diameter

When using a non-geared extruder on a 0.25mm diameter nozzle, I was experiencing what looked like massive under extrusion. I checked my E-steps and tweaked my slicer settings multiple times but could not fix the problem.

What was occurring was minor skips in the extruder motor, and minor grinding of the filament.



This happened because a non-geared extruder does not have enough torque to push through such a small nozzle due to bottlenecking.

You will even see on E3D's website that they say a geared extruder is required to print using their extremely fine diameter nozzles. I can't stress enough how much I do not like the plastic non-geared extruders that come on inexpensive machines. If you print with small diameter nozzles, you will definitely want to upgrade to a geared extruder, such as those made by Bondtech, E3D, DropEffect, and many others.

Check and fix your E-steps

For most under and over extrusion issues, you will want to check and calibrate your E-steps. To do this is actually quite simple.

You will want to start off by measuring out 100mm of filament. You can actually measure out even more for a more precise readout – you will just have to account for that in the calculations below. I prefer to use White PLA because it is the easiest to write on, has a low printing temperature, extrudes easily, and is cheapest - though you could use any material you have at your disposal.

You can do this in whatever method is easiest for you. I found it easiest to measure this 100mm when the filament is already fed into the extruder. You can also do this on a desk before feeding, but 3.00mm filament, and 1.75 near the end of its spool, are quite hard to keep from rolling back up.

Be as precise as you can by using a fine tip sharpie and holding the material as straight as possible. Use calipers if you have them at your disposal. After heating your hotend, you then want to push the filament down until the lowest dot you made lines up with the top of your extruder, or somewhere else you can easily line up the starting point (because you will need to compare it to where it finishes). If you line this dot up to right where the filament is fed in, you won't be able to see if you over extruded, since your finishing dot will now be inside your hotend/extruder.

The next thing you will want to do is to tell your printer to extrude 100mm. This is done with a simple G-code command in your terminal.

If you normally print via SD card you will need to hook up to a computer for this. If you print via Octoprint or a similar online program, you can send the G-code commands from their terminals. I personally have most of my printers hooked up to Octoprint via a Raspberry Pi, but this likely will not be common for those of you who are new to 3D printing.

If you do not have your printer connected to Octoprint, you will want to download Repetier Host to your computer, unless you have another software you can use when your printer is hardwired. Normally just choosing “auto” for your baud rate will work, but if you have difficulty with your particular printer, just search online for your printer name and “baud rate”. When hooked up to Repetier Host, or whatever program you use to control your machine, and with your hotend hot, you will want to give your machine the command:

G92 E0

This sets your extruder to 0. Next you will want to give either the command:

For 3.00mm Filament: G1 E100 F30

or

For 1.75mm Filament: G1 E100 F60

The “F” in this equation is just referring to the speed, so you don’t need to use those exact numbers, this is just what I use to make sure we don’t get any extruder motor skips and we are feeding exactly what we think we are feeding. You can always go slower or lower with this number, it will just take a longer amount of time to feed your filament. This will tell your extruder to feed 100mm, and is why it was important you lined up your starting dot with either the top of your extruder or something else that is easy for you to compare to.

Once your extruder has finished you will want to mark your filament at the same spot you lined up your original dot (top of extruder in my examples). If your 100mm dot lines up perfectly, then your E-steps are right on - but even 1mm means that your printer is extruding incorrectly by 1%.

After marking where 100mm actually was, you will want to compare it to where you measured 100mm to be at the beginning of this process. If higher on the filament, your printer is over extruding, if lower on the filament, your printer is under extruding.

After measuring this difference you will want to write down somewhere how much your extruder actually fed. If your printer over extruded by 2.1mm, you will want to mark down 102.1mm. If it under extruded by 2.1mm, you will want to mark down 97.9mm. You will need this number later on.

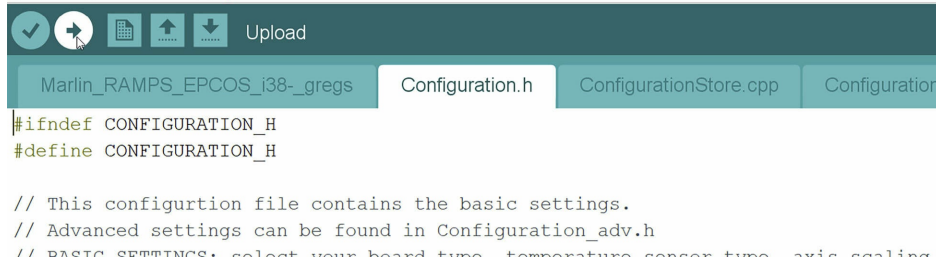
The next step in this process is to determine what your current E-steps are. You can do this by either checking the firmware for your machine, by going into the “Motion” section of your LCD screen if available, or just by giving it the command “**M503**” in your printer terminal. This means you would type “M503” into Octoprint, Repetier, or whatever software you decided to use that you can send G-code. This “M503” will spit out all of your printer settings and you will need to scroll up a bit to see what your printer has set for its E-steps.

Non-gearbed extruders have E-steps of around 90, while Greg’s wade and other geared extruders can have E-steps of 500 or more. Something like the Titan has a starting point of 420. The Hemera is closer to 400. If you have an extruder from a popular manufacturer, they will list what their standard starting point for E-steps should be. It will help to set your E-steps to this manufacturer recommended number before running your E-step test if you are swapping extruders.

If you are checking in the firmware that you use to flash your machine, you will want to open it up. I actually rarely do this because sending G-code commands is much easier. While in Marlin you will go to the “Configuration.h” tab and scroll all the way down to where it says “DEFAULT_AXIS_STEPS_PER_UNIT”, with E-steps being the 4th and final number (if using one extruder). The X, Y, and Z steps should never be changed and are a calculation based off of the parts you are using.

Marlin_RAMPS_EPCOS_i38_gregs - Configuration.h | Arduino 1.8.6 (Windows Store 1.8.14.0)

File Edit Sketch Tools Help



```
#ifndef CONFIGURATION_H
#define CONFIGURATION_H

// This configuration file contains the basic settings.
// Advanced settings can be found in Configuration_adv.h
// BASIC SETTINGS: select your board type, temperature sensor type, axis scaling
```

The easier method is to just type “M503” into your terminal to be given a readout of what your current E-steps are.

After running your 100mm feed out test, you then take your current E-steps number that you found via your firmware or via the “M503” command and multiply it by 100 (or the amount you were attempting to extrude if you decided to test 200mm instead). You will then divide this new number by the number you wrote down earlier.

For example, if your current E-steps are 90.5 as shown in Marlin above, you will multiply it by 100 to get 9050. We will then divide 9050 by how much you actually extruded earlier. So if you extruded 102.1mm (over extrusion), you will take 9050 and divide it by 102.1 to get 88.64.

$$90.5 \times 100 = 9050$$

$$9050 \div 102.1 = 88.64$$

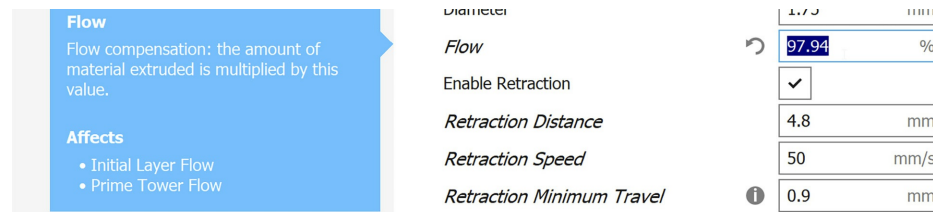
88.64 in this above example would be your new E-steps. As you can tell it is lower than it was before, because in this example you were correcting for over extrusion.

You will now set your E-steps. You can do this through your terminal, EEPROM, or by flashing your firmware. If you are going to do this through your terminal, which is my preferred method, you will want to give the M92 command, by typing “**M92 E88.64**”. You will then want to type “**M500**” in order to save these settings. Without typing “M500”, the E-steps will be reset when turning off your machine. Remember to have an SD card in your machine when doing this “M500”, since some printers require it to save properly.

While you can set this number on LCD screens under the “Motion” section on some printers, it will only save permanently if you have the option to save your settings after doing so, just as with typing “M500” in the example above. Otherwise your E-steps will reset once you turn your machine off. Unfortunately many printers made today actually do not have this options, meaning you will need to do it via G-code commands or flashing firmware. Thomas Sanladerer has a great older tutorial video going over all of this on his channel which you can find by searching “calibrating your extruder” on YouTube. Thomas really knows his stuff and I suggest to everyone that they follow what he does, since he is my personal favorite YouTuber on 3D printing information. That said, there are newer videos on YouTube which will help walk you through this with particular machines, in case you are having problems.

Changing Flow % in slicing software

If you have a machine that will not allow you to use Marlin and cannot save the settings using the M500 command, you could literally just change the Flow % in Cura. I do not suggest this though since sending G-code commands should work with your printer. Remember to have an SD card in your printer if the M500 command is not working.



This is not ideal, but can get the job done if needed. Essentially, you will not need to know any E-steps, you will just need to know the percentage that you over or under extruded in your test you performed earlier. If your extruder fed out 102.1mm, then you would divide 100 by 102.1 to be read out .9794. You can then set your flow percentage to 97.94%.

Summary of Fixes and Precautions

- Check if you have your slicer set to the correct filament diameter.
- Replace nozzle if degraded or if you have done countless prints on a standard brass nozzle.
- Check the tension on extruder idler.
- Confirm your extruder gear is attached firmly to your stepper motor shaft and that there is no free play.
- If your extruder is “clicking” or experiencing skips – check the “Extruder Motor Skipping” chapter.
- Confirm you are giving enough torque for the nozzle diameter you are using.
- Check your E-steps and set them to the proper amount via the tests explained in this chapter.

Parts Being Knocked Over

Your part being knocked off the build plate can happen quite frequently, especially when dealing with tall, skinny prints on Cartesian machines.

Proper bed adhesion

Goal number 1 when it comes to having parts not being knocked over is having the proper bed adhesion. This includes knowing the right mixture or bed sheet for the material you are using, the correct bed temperature and brim/raft application, as well as having the proper z-height with a level build plate.

To review this further, make sure you check the “Bed Adhesion”, “Unlevelled Build Plate”, and “Z-Height” chapters. If any part of your print is too far from the build plate, you will be very susceptible to your part being knocked over.

Adding a Z-hop

As mentioned elsewhere in the book, a Z-hop refers to the printer head moving up while travelling. The printer head (or build plate when working with a CoreXY machine) will have the nozzle move away from the print by the determined amount in your slicer settings. This will mean that the nozzle will not be running into your print when travelling, reducing the odds of it being knocked over.

I always have my Z-hop height set to the layer height of my print. I have just found this to work best, because if you have a Z-hop set lower than the layer height, you run the risk of still bumping into the top of your print. I increase the Z-hop larger than my layer height when printing at large layer heights (over 0.4mm).

I have actually found though that some inexpensive printers have difficulty with Z-hop, and it can add artifacts that you don't want, such as covered in the "Layer Bulges" chapter. If you are experiencing any ugliness such as that in your inexpensive printer, then you may want to just not use a z-hop and go onto the next setting.

Avoid parts when traveling

Along with the ability to add a Z-hop, Cura also allows you to “Avoid Printed Parts When Travelling” and “Avoid Supports When Travelling”. If you are experiencing any difficulty with Z-hop, go ahead and check those two boxes in the “Travel” section. This will add to your print time, since your tool path will not be the shortest one possible, but it is definitely beneficial if you are not using a Z-hop.

In fact, I have been so accustomed to using a Z-hop, I never really realized that it isn’t a great method to use on inexpensive printer, excluding Deltas. A well-built Voron or similar style machine may achieve a Z-hop without any issues, but it is recommended to not use it on inexpensive Cartesian machines with a single leadscrew. This is only something I recently realized after having a lot of difficulty with Z-hop on my Ender 3 V2.

Travel	
Enable Retraction	☒
Retraction Distance	1.5 mm
Retraction Speed	30.0 mm/s
Retraction Minimum Travel	0.9 mm
Combing Mode	All
Avoid Printed Parts When Traveling	☒
Avoid Supports When Traveling	☒
Z Hop When Retracted	☐

Turn combing off

When combing is on, you get the option to avoid printed parts when traveling. This isn't always the best option though, since it will depend on just how large your layer heights are. It seems that when I use large layer heights, even with avoid printed parts when travelling checked, my nozzle will still run into the infill. For this you will want to turn combing off. It may add to your retraction headaches, but the printer will always perform a Z-hop when travelling and will avoid your printed part.

I have combing on for any print that is 0.3mm layer heights or lower, but anything higher I will have it turned off.

When combing is off and you are printing with large layer lines, it should also be accompanied with an infill pattern that only goes in one direction per layer (such as "lines"). That is something I learned when printing with the E3D SuperVolcano at 1mm layer heights. Those prints were impossible not to have my nozzle run into the infill if I didn't do that.

Working with very thin tall prints

This is also covered in the “Z-Axis Wobble” chapter, but essentially a very tall skinny part is far more likely to wobble and get knocked over during a long print. This should not be an issue on CoreXY machines where the build plate only moves downward, but on Cartesian machines where the build plate rattles back and forth, the very top of a tall skinny print will start to sway back and forth.

This swaying will not only cause ugly Z-wobble prints, but it can cause a print to be knocked over. This is because the nozzle may try and start the print slightly to the side of where the top of the print swayed to. Get this happening a couple of times and this part will easily fall over.

This is particularly noticeable when working with thin support material. Often a part will require a very tall support structure, but may not need to cover a large surface area. These thin towers can fall over before they get to where you need them to be.

Unfortunately, when working with a Cartesian machine, there isn't much that can be done to prevent this. Of course you need to have a proper brim for great bed adhesion, but if a part is extremely thin and tall, there will be swaying. There are essentially two things you can do in this scenario – cut the part into two sections or add further scaffolding to help anchor.

There have been some thin parts I have printed in the past that I was just forced to slice in half and glue together post printing. This is clearly not ideal for mechanical parts, but anything this thin will be flimsy regardless.

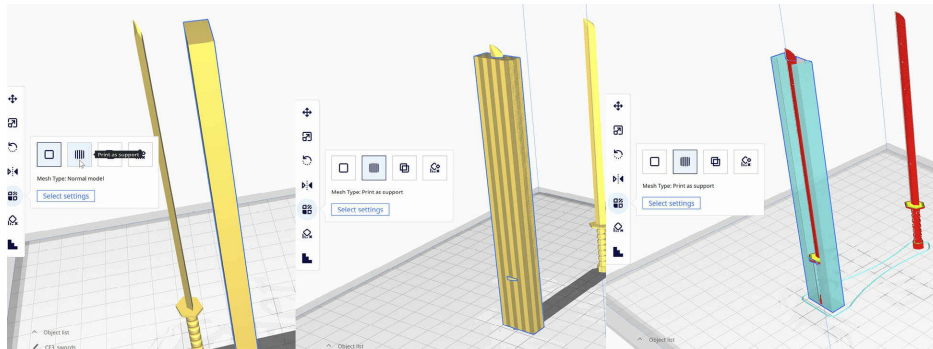
You can also add manual support structures to the model, anchoring the part to the build plate every so often, to make sure no swaying occurs. If you are just building a thin tower, you will need support structures attaching to the side. This can be done in programs such as Cura.

Anchoring Prints

I am sure there are better ways to do this, but the easiest way I know of would be to anchor your print in Cura. I have rarely needed this, but it definitely helps with a tall, skinny print wobbling back and forth. This is also explained in the “Z-Axis Wobble” chapter, as well as the “Cura Tricks” chapter.

Below is an example of two skinny swords from a Deadpool print that I made for my YouTube channel. When not adding any anchors, my Cartesian machine would wobble the build plate back and forth and cause the top half of these swords to look extremely ugly (if they didn’t just get knocked off).

Cura allows you to bring in a second model that intersects with your main print. They also allow you to print a part entirely as support. This means you can drag in a second object that acts only as support for your main structure.



This rectangle in the example above is thin, so it won’t take up too much material, yet it will extend the anchoring for the sword (I added a second sword to compare how it will slice). After bringing in a shape that will work for your model, you can choose the model and click “Per Model Settings” and then “Print as Support”

After turning the shape into “Print as support”, you can then drag it over your tall, skinny print.

As you can see in “Layer Mode” this entire shape is now support structure that can help to anchor your tall skinny print to help prevent this wobbling back and forth.

As mentioned – there are many other ways to do this, this is just the simplest way I know of since it allows you to do this right in your slicing software.

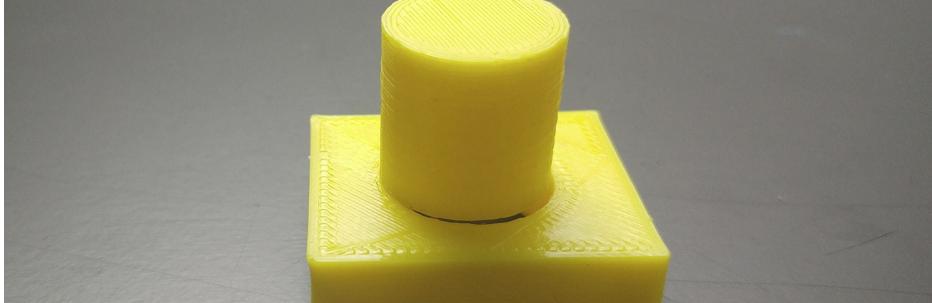
This process is explained in detail as well on my video titled “Combining Parts and Multiple Processes on Cura” if you would like to see it in action along with before and after shots.

Summary of Fixes and Precautions

- Ensure you have the proper bed adhesion – including having a level build plate with the proper starting Z-height.
- Add a Z-hop (normally the same distance as your layer heights).
- It is preferred to not use a Z-hop on inexpensive machines and instead utilize the ability to avoid parts and support when printing in Cura's slicing software.
- Turn off combing and use “lines” as pattern infill when printing with very large layer heights.
- Plan ahead when working with a tall and skinny print. You may need to cut the part in half or add anchoring to make sure the part doesn't wobble back and forth.

Parts Not Mating Together

This issue can be as minor as when you print two parts that are meant to fit together and they don't without some sanding, to as far as one or multiple axes being off by over 100% in scaling.

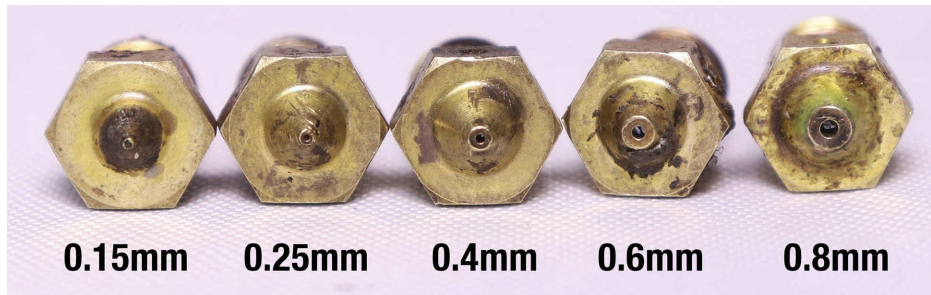


You will have to diagnose how bad this problem is. If it is minor, proceed with the next couple of steps.

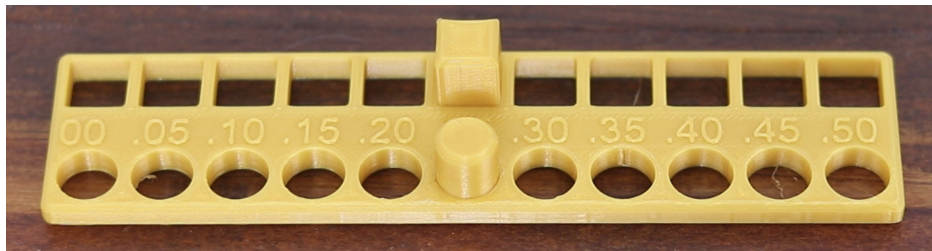
Confirm you factored in the appropriate tolerances

There are clearly tolerances involved with 3D printing, and they can vary drastically on the nozzle diameter and layer heights of your particular print. While there may be sources that say the tolerances are tighter than the numbers I am saying, these are what I use and what has worked for me throughout all of my prints.

The tolerances of the print is the layer height you are using in the Z-direction, and roughly half the diameter of your nozzle in the X/Y direction. So, as an example, if you were printing 0.1mm layer heights on a 0.4mm nozzle, your tolerances will be roughly 0.1mm in the Z-direction and 0.2mm in the XY direction.



While that is a general rule of thumb, you may actually be able to achieve tolerances tighter than this on your particular machine depending on how everything is set up. If you are under extruding, you may be able to achieve even tighter tolerances, though your parts may be structurally weaker than they should be. You should find out your exact tolerances by printing a tolerance test – like the one below designed by A_Str8 on Thingiverse (though there are many out there that should accomplish the same idea).



Since 3D printing is additive, it will almost always err on the side of adding more material than less material. This means that parts will be slightly larger and holes will be slightly tighter.

So, if you were to design two parts that mate perfectly together without any clearance, no matter how good your settings are, you will likely not be able to

fit the printed pieces together without a lot of sanding.

I always suggest parts that need to be perfect to their dimensions, or ones that need to mate together, have the size of their holes increased, and the overall size of the part decreased. The amount you should factor in should be based off of the tolerance test I suggest you print a couple paragraphs earlier.

For instance, using the same example as above, if you are printing at 0.1mm layer heights on a 0.4mm nozzle, you will want to increase the diameter of holes being printed in the XY direction by 0.2mm, and decrease the size of the part in the XY direction by 0.2mm (and 0.1mm in the Z direction).

With this tolerance, parts will mate tightly together. If you would like the fit to be a little loose, you will want to increase this to 0.3mm for the XY tolerances in the above example.

Based off of the photo I show above, I am able to achieve tolerances of 0.25mm in the XY direction for a really tight fit, but 0.35mm for a looser fit. So you need to factor in the corresponding clearances in your part depending if you want a tight or loose fit.

This is another reason printing in a high resolution on a fine nozzle is beneficial for printing a part with accurate dimensions, but will of course result in a much longer print time.

Replace your Nozzle

Your nozzle will be degraded over time, especially when using one made of brass. Abrasive materials at hot temperatures will slowly make the diameter larger than what you think it is.

You can see some example prints and more explanation in my YouTube video, “The Importance of Replacing Nozzles”.

While you are trying to lay down 0.4mm layer lines with a 0.4mm nozzle, your nozzle may actually be closer to 0.6mm in diameter after it has been degraded. This can lead to ugly parts, and can definitely lead to your parts not mating together properly. If you are using a brass nozzle, make sure you replace it every couple hundred hours of printing. I definitely suggest upgrading to a hardened steel nozzle so this does not become an issue nearly as quickly.

Check to see if you are over extruding

You will definitely want to visit the “Over and Under Extrusion” chapter in this book if you factored in tolerances but parts are still coming out slightly too large.

Be sure you are as accurate as possible when checking your E-steps because even a slight over extrusion can cause problems when you are trying to print two parts that are meant fit together accurately.

Many people suggest setting your E-steps to 98% of the number you are fed out, since a very minor amount of under extrusion should help with mating parts together. I personally do not do this, but you can always reduce the “Flow %” in your slicer to 98% or 99% for parts that really must fit together.

Make sure your material hasn't absorbed moisture

Materials left out not vacuum sealed or without a dehumidifier will undoubtedly absorb moisture. When this occurs, you will experience a myriad of printing issues, one of which being having trouble mating parts together.

You can read more about this issue in the “Stripped Filament” chapter.

Tighten belts

As mentioned elsewhere in this book, it is possible to over tighten your belts, but you will really have to be trying to do that. If you are experiencing parts that are not to the correct dimensions, your belts may be too loose.

You do not want a lot of slack in your belts because not only will quality of your parts decrease, you can experience actual dimensional issues. This happens slowly over time on just about every machine I have used. I have actually experienced a part printed on one machine not mating with a part printed on another.

It turned out that tightening the belts fixed the issue immediately. This is another reason having a way to easily tighten your belts will be extremely useful and necessary for practical preventative maintenance.

Confirm stepper pulley has proper amount of teeth

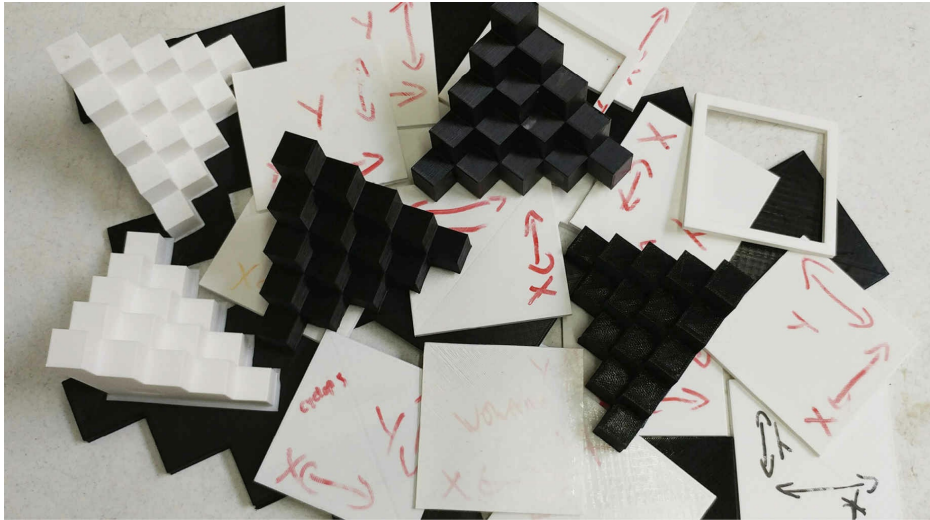
This is a strange one and should only occur if you recently changed the pulley attached to your stepper motor. This happened to me once, and it took me about 3 prints of confusion to realize what was going on.

If your stepper pulley has the incorrect amount of teeth for your machine, your scaling for that dimension will be very far off. From what I can tell, the vast majority of FDM machines use 20 teeth pulleys for their stepper motors, but you should double check before replacing any on your printer.

Read the “Settings Issues” chapter

If you have confirmed everything above, and parts are not mating together, make sure you read the “Settings Issues” chapter. You would be amazed just how much tweaking your settings can affect the quality of your print. Along with making sure you don’t have your flow rate over 100%, minor fluctuations in temperature for specific materials can affect the viscosity which can cause a print to ooze more than it should. This will result in a part that is not to the proper dimensions.

Print a large calibration cube and check XYZ steps/mm



This really should not be needed if everything is built properly and you are using the correct firmware for your machine - but it may be smart to do on very large machines that you built yourself.

Find a calibration cube on Thingiverse, or create your own that is large and uses the least amount of filament as possible. This can be a hollowed out cube that is only shells and has not top or bottom, or whatever you can think of that has specific dimensions of at least 100mm. If you have the filament, space, and time, you should go even larger. This process would then be the same as when you checked your E-steps.

I cannot stress enough how important it is to check your E-steps and to check everything else described above before printing your calibration part. There is no point changing the X, Y, or Z steps if a belt is loose or if you are over extruding. Your X, Y, and Z-steps should be a calculation based off of the parts you are using – and there is no reason to tweak this UNLESS you built the machine yourself and you were wrong with your calculations in flashing firmware.

Find your X, Y and Z steps per mm in your firmware – they are located directly to the left of the E-steps/mm (as explained in the “Over and Under Extrusion” chapter). You can also get them by sending the printer a “M503” command. Take that number and multiply it by the number of mm your calibration print should be in that dimension (if printing a 100mm cube, multiply by 100). Pull out your calipers and measure your print in that direction for what it actually printed.

Make sure you do not scrape off your print before marking or remembering which direction is which. Divide that newly found number by the actual number read out by the calipers. This is your new steps per mm for that dimension.

As mentioned, if you have a printer that is using the proper firmware and parts, with settings that are correct for the material you are using, this should not be needed. I never do this on my machines because everything is just based off of calculations.

Summary of Fixes and Precautions

- Make sure you factored in clearances for parts that need to be mated together or precise. These tolerances have to do with your layer heights and nozzle diameter.
- Print a tolerance test to know what your clearances should be.
- Replace any worn out nozzle.
- Make sure you are not over extruding by checking the E-steps.
- Tighten all loose belts.
- For parts that are far off in their accuracy, confirm you have the proper amount of teeth for your stepper motor pulley for your machine/firmware.
- Read the “Settings Issues” chapter since many issues in minor dimensional accuracy have to do with having the proper settings for the material you are using.
- For large machines, or machines you do not have the firmware for, you may want to check the X,Y and Z steps/mm by printing a very large calibration cube. Only do so after confirming all of the above.

Patterns in Outer Surface

I had personally always thought that patterns in the outer surface of your print was directly correlated to “ghosting,” though I have recently been exposed to a completely new reason this could be occurring. If you are seeing a repeating pattern in your outer surface unrelated to your design, then keep reading.

By reading this book and my previous edition, I am sure you know I am a big fan of dual-gear extruders since they offer more grip and a more precise extrusion, meaning you can print more material options more accurately.

That was until I saw a video by Mihai Stanimir of MihaiDesigns on a YouTube called “What I’ve learned about dual gear extruders and how to patch them, maybe.”

I sincerely suggest watching that video from beginning to end, but essentially the main reoccurring factor in this issue was a dual drive extruder.

Dual drive/geared extruder issues

As Mihai covers in his examination video, it seems that the dual geared extruder can be the culprit for this outer pattern/wood look.

Essentially, when the two hobbed gears that grab the filament are closed completely, the gears will spin just fine and interlock as they should. The problem comes from when you push in filament, since the hobbed sections will be pushed slightly apart from each other to accommodate for the width of the filament. This then creates a small gap between the two gears that are supposed to turn in unison and push the filament out.

This minor gap between the gears can cause one gear to turn a slight amount before turning the second gear that is instantly pulled back when it catches. This isn't enough to fail a print, since we are talking about a mm or less of rotational movement. That said, it seems that this starts to cause a pattern in the outer edge of your print that repeats in a recognizable fashion.

Mihai did many tests, and the thing that seemed to remove this outer surface pattern was turning his dual geared extruder into a single geared one that pushes against a bearing. This is the standard way extruders have been built, and is definitely not preferred when you want a dual geared extruder.

As of writing this, this seems to be the only real solution. Personally, I have 2 Hemera dual drive extruders; an OmniaDrop 2.0, plenty of Bondtechs, and one BIQU H2 dual drive extruder that are all running just about daily. I printed the test that Mihai includes (MihaiDesigns.com/Pages/Inconsistent-Extrusion-Test), and none of my tests showed this pattern. I really tried, and it was impossible for me to duplicate. I did not get approval from Mihai to use his photo, which is why there is no photo of this in the diagnosing section or in this chapter. Make sure to check the link included and watch his video to see if you are experiencing this.

It is difficult for me to say why this is occurring for some dual geared extruders and not for others, but it may play into my point that it is always important to buy name-brand goods rather than knockoffs. E3D's quality control is top notch as is their customer support, so I can only suggest going with someone like them. If you see this wavy outside pattern and it is coming from a name brand dual geared extruder such as E3D Hemera, I would suggest emailing them directly to let them know of the issue in hopes you can get a replacement. This would not be nearly as simple if you are buying an off-brand Bondtech with minimal to zero customer support and quality control.

Regardless of whether or not you have a name brand extruder though, Mihai

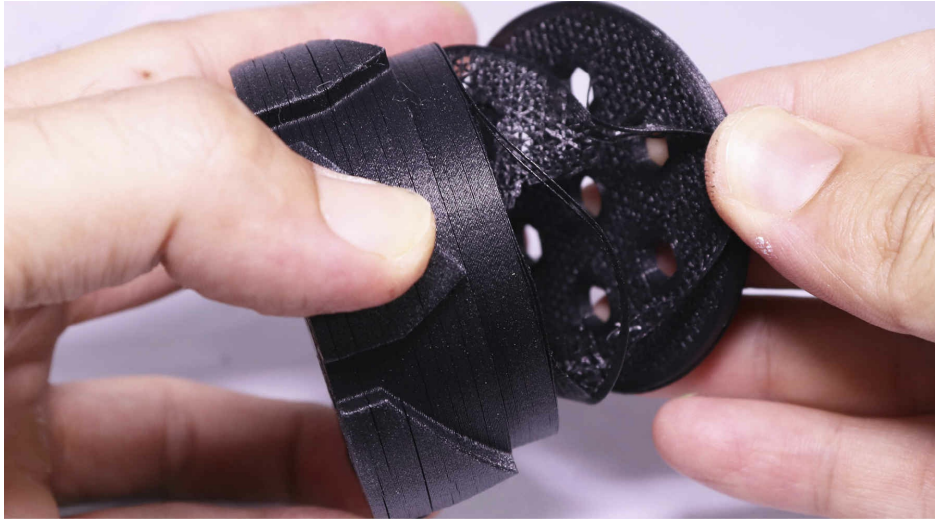
brings up a very valid issue that needs to be addressed. As he suggests in his video, extruder companies could swap over to more of a herringbone gear design which should eliminate or mitigate this issue. There are likely quite a few different gear designs that could work.

Summary of Fixes and Precautions

- This issue doesn't have too many explanations, since Mihai proved that it is directly related to a dual geared extruder.
- Disassemble your extruder before buying a new one or finding replacements in order to make sure there isn't anything noticeably wrong.
- Buy only name brand extruders. I am sure this issue is far more common with knockoffs.
- If you have a name brand extruder and this is happening, I would suggest reaching out to the manufacturer. I am not sure if they will do anything about it, but this is definitely a flaw in the dual gear design.
- You can always swap the second gear for a bearing in order to turn your extruder back into a single gear. This is not ideal but will eliminate this issue.
- If you tried contacting the manufacturer to no avail and can't replace the second gear with a bearing, the only thing I can really suggest is purchasing or building a new extruder.

Poor Layer Adhesion

Having strong layer adhesion is not only mandatory for watertight parts, but it is needed for clean, strong prints. If your individual layers do not stick together well, you are bound for a part that will break and peel apart.



Understand the material being used

Each material requires its own settings, including temperatures and speed. You need to make sure that you are using the proper settings for the type of material you are using.

Carbon fiber reinforced blends are more likely to have poor layer adhesion due to their properties. Polymaker is currently working on carbon fiber reinforced blends that increase this layer adhesion, but it is going to be a problem with most of these types of materials.

Refer to the “Materials Sciences”, as well as the manufacturers print settings, before moving forward in trying to fix this problem.

Increase the extrusion temperature

One of the most common reasons poor layer adhesion occurs is because you are printing at too low of an extrusion temperature.

Following the manufacturer guidelines is normally a good way to prevent printing at too low of a temperature, but a few times I have had to go above these recommended settings to make sure I had a strong enough layer adhesion. This is particularly true with flexible filaments if I am printing them fast. I have had to go as high as 25 degrees above the recommended range on some flexible materials just because I was printing them much faster than suggested due to having an upgraded extruder setup.

The photo at the beginning of this chapter was a failed print I had when printing in NylonX by MatterHackers. They recommend 250 – 265 degrees Celcius for printing, so I went ahead and tried 250 degrees. I was left with the part you see above, something with far too weak of layer adhesion. After upping this to 265 degrees the part printed much better.

This problem was also increased because NylonX is a carbon fiber reinforced nylon, and as mentioned earlier in this chapter, carbon fiber reinforced materials are more likely to experience poor layer adhesion.

Try slightly increasing your extrusion temperature to see if it helps with this problem.

Print with a larger diameter nozzle

As covered in the “Material Science” chapter, you increase your layer adhesion by increasing the amount of entanglements between the layers. One way to increase this layer adhesion is to increase the surface area of your nozzle. Using larger diameter nozzles will help quite a lot with proper layer adhesion.

Slower print speeds

Similar to the explanation for using a larger diameter nozzle, you can also slow your print speeds down so that the nozzle is in contact with the layer below it for a longer period of time. With a larger nozzle and slower print speeds, the amount of entanglements between your layers will increase, meaning you will have stronger layer adhesion.

Whenever I am printing with something like ABS or ASA, I prefer a 0.6mm nozzle and I print very slow and very hot. I actually print ABS around 260°C and at around 35mm/s. This is much hotter than most manufacturers say ABS should be printed at, and much slower than many makers advertise. But slow and hot will really help with making sure your parts have very strong layer adhesion.

Under Extrusion

Another reason for poor layer adhesion is under extrusion itself. If your extruder is depositing less filament than it thinks it is, you are bound to have weak layer adhesion.

Confirm you have the proper E-steps set by referring to the “Over and Under Extrusion” chapter in this book.

This could also be from temporary under extrusion, and I cover a lot of tips for that in the “Missing Layers and Holes in Prints” chapter. You will essentially want to make sure that your extruder gear is attached tightly to your stepper motor shaft and not have any free play, though you should read that chapter in full to get a detailed explanation.

Not enough torque

As with the “Extruder Stepper Skipping” chapter, you could be working with a stepper motor/extruder setup that does not have enough torque. If your extruder motor skips, you will essentially be left with an under extruded or poor layer adhesion print.

The best way to remedy this is to upgrade to a geared extruder, if you haven’t already. There are many types of geared extruders, as covered in this book, and my favorites are made by E3D and Bondtech.

Turn off active cooling fan

While having your active cooling fan turned on will benefit the majority of prints and materials with their surface quality, many filaments require you keep this active cooling fan off for proper strength and layer adhesion.

Another reason the print from the photo in the beginning of this chapter failed was that I kept my active cooling fan on. It seems that MatterHackers states their NylonX should be printed without any active cooling fan.

This not only helps to prevent warping, but will also help to achieve strong layer adhesion. Generally, the higher the heat capacity and density of the polymer, the more beneficial a cooling fan will be. For lower density polymers such as ABS, HIPS, etc. that are below 1.2g/cc, the fan is typically recommended to be turned off.

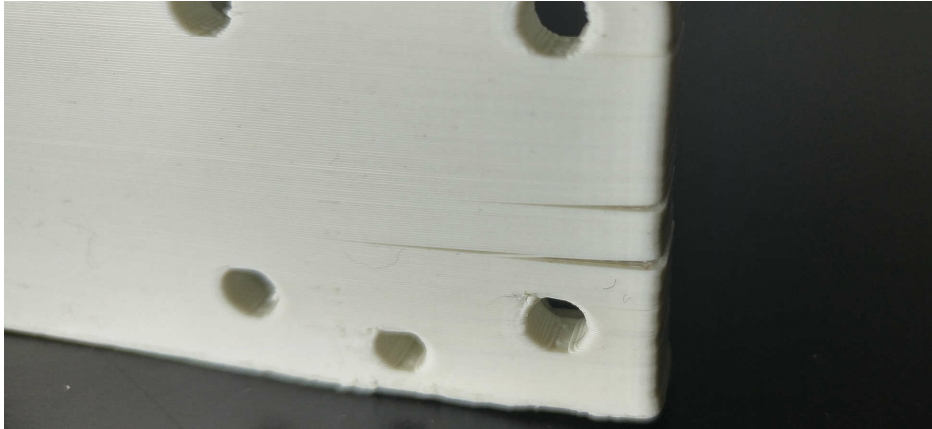
Make sure to check the manufacturers recommended settings for the particular material you are using, since many materials will benefit from increased layer adhesion when there is no active cooling fan.

Make sure material is stored properly

As with many other problems, make sure your material is stored properly. Old, wet material, or poorly made, off-tolerance filament can lead to issues that are very difficult to diagnose. If you consistently get poor layer adhesion from one spool regardless of what you do, try using a different spool. If the issue does not continue, it is likely problems with that particular material.

Delamination

I personally consider delamination a different problem than just poor layer adhesion. You can have extremely strong layer adhesion but still experience delamination when working with high warping parts.



Because of this – I cover this specific issue in the “Warping” chapter, since delamination is far more a symptom of high warping materials rather than just poor layer adhesion.

Summary of Fixes and Precautions

- Read the “Material Sciences” chapter.
- Check the manufacturer recommended print settings. Do not go faster or change the temperature outside their ranges to start your tests.
- Switch from using a carbon fiber reinforced material if continually experiencing issues, since carbon fiber reinforced materials are going to have more issues with layer adhesion than other types of filaments.
- If experiencing poor layer adhesion while staying within the recommended settings, try increasing the extrusion temperature slightly. I actually print ABS and ASA much hotter than most recommend.
- Use a larger diameter nozzle.
- Slow your print speeds down.
- Check E-steps and confirm you are not under extruding.
- Confirm you have enough torque and that your extruder motor is not skipping.
- Turn off active cooling fan if the particular material does not call for it.
- Make sure there is no issues with the material you are using.
- If experiencing delamination – check the “Warping” chapter.

Problems with “Power Loss Recovery”

One great upgrade that has appeared over the last year or two is “power loss recovery.” This means that if you happen to lose power or your printer restarts mid print, you won’t be left with wasted material and time. The printer will ask to resume where it left off, and hopefully continue exactly where the power loss occurred.

The problem is that some manufacturers seem to not have implemented this feature perfectly and it has led to two different problems that I have seen over the last year.

Print pauses after each layer

This has only happened to me with one printer – the FLSUN Super Racer. This Delta printer is advertised as having fast printing, and that is definitely true. The issue came into play when the print would actually take much longer than the slicer said, because the printer would do a little pause in-between each layer.

I couldn't figure this out and thought it had to do with the "Minimum Layer Time" setting on Cura, but that wasn't it. Modbot, a fellow YouTuber that I highly recommend checking out, was actually the one to figure out that this had to do with a malfunctioning power loss recovery.

The printer seems to save where it is after each layer in case of a power loss, and then start the next layer. Unfortunately this process takes a lot longer than it should, leading to blobs and a print that takes much longer than the slicer states which kind of defeats the purpose of "fast printing."

Right now the only method I have to fix this would be to turn off power loss recovery entirely on the printer. You can do this in the settings on the printer if your particular printer allows for it, otherwise you will need to add a script to your "Start g-code." This code would be "M413 S0".

You will now no longer have power loss recovery, but it will prevent your printer from pausing after each layer is completed.

Issues with “Vase Mode” or “Spiralize the Outer Contour”

This particular issue has only occurred for me on the new Ender 7 by Creality, and seems to be a bug they may be able to fix before you ever experience it. That said, it is worth mentioning in case you have this issue as well.

Whenever I tried to print in “vase mode,” where the printer just spiralized one wall to create a hollow print, my Ender 7 would pause, retract, and move to another small section and repeat the process. This means not only was my wall not spiralized, it had a ton of holes and took about 4x as long to print as the standard settings said.

This was fixed by either turning off the power loss recovery, or by a weird trick of turning it off at the beginning of the start g-code, and then turning it back on at the bottom of the start g-code.

You may also experience blobs on your spiralized print from power loss recovery turned on, but that is more of a bug explained in the previous section, where your printer pauses for a small amount of time after each layer is completed, which obviously defeats the purpose of vase mode.

Update your firmware

This seems to be a software issue since it is not true across all power loss recovery printers. Your particular printer manufacturer may have fixed this issue with updated firmware. Before giving up entirely on this added feature, see if you can update your firmware to remove the issue by searching for your particular printer.

If not available for your firmware, unfortunately you will lose the ability to start a failed print when it is turned off, but these blobs and ugly features will be mitigated.

Running Out of Filament

This problem is by far the easiest to diagnose but also one of the most frustrating when it occurs. You can think you have enough filament for that 400 gram print, when 20 hours in, with only 10 layers left, you run out of material.

You can avoid this by taking these precautions:

Weigh an empty spool

It is always good to have the weight in grams of an empty spool for the filament manufacturer you are using. These do have tolerances, but it is a good starting point.

After you have the weight of an empty spool you can then weigh the spool you are about to use for your next print. Subtract the weight of the empty spool and you should have a rough estimate of how many grams of the material are left. Make sure to provide a buffer of at least 20 grams to account for tolerances in the spool itself.

Pause at layer height if you know you will run out

If you are going to start a print with a spool that you know does not have enough filament to complete, you can add a “Pause at Layer Height” to the slice of your model. To do this on Cura you would go to “Extensions” at the top, then scroll over “Post-Processing” and click on “Modify G-Code”.

You would then add the script “Pause at Height” from the list. You can then choose the exact layer or height that you would like your printer to pause so that you can swap to a new spool. This is especially useful for very large prints and you don’t have a full spool available.

You can also use this feature if you would like the top portion of your print to be a different color than the bottom.

Understand the density of the material you will be printing with

When you use a slicing program it will give you an estimate of the amount of material it will be using. If it gives you this number in grams, it will not be accurate if you are using a material that is different than is in your machine settings. If you are given this estimation without having to set anything up, it is likely based on the density of PLA.

If you are given the estimation in meters and not grams, you will have to do a minor calculation to find out the estimation in grams for your material.

PLA is 1.25 grams per cubic cm. If you were using 1.75mm diameter filament, one meter of filament would be 2.41 cubic cm in volume.

1 meter of PLA 1.75mm filament would then be equal to 2.41×1.25 , or 3.0125 grams. A 1,000 gram spool of PLA should be roughly 331 meters.

Using that 2.41 cubic cm in volume for 1.75 filament, you can use the data below to figure out how many grams of material your print expects to use.

Density of material

PLA: 1.25 g/ccm - 3.0125 grams per meter of 1.75mm filament

ABS: 1.04 g/ccm - 2.5064 grams per meter of 1.75mm filament

PET: 1.38 g/ccm - 3.3258 grams per meter of 1.75mm filament

Most Nylons: 1.13 g/ccm - 2.7233 grams per meter of 1.75mm filament

The same is true for 2.85mm filament, you would just use 6.38 cubic centimeters as the volume per one meter of filament.

Use a filament runout sensor

Many printers now feature a filament runout sensor. You can purchase one as well, it would just require you to do some tweaks to your firmware. These sensors are very inexpensive and work in a way that they pause your print when filament is no longer running through it. This means if your print runs out of filament while you are away from the machine, you will come back to a print that is paused with the nozzle off of the print. You can then change the material to a new spool, and click “resume”.

Summary of Fixes and Precautions

- Weigh your spool before starting print.
- Add settings to pause at a layer height to allow you to switch filament if you know your spool does not have enough material.
- Know the density of the material you are using to calculate the estimated amount of grams for your print.
- Use a filament runout sensor.

Settings Issues

This is a very vague chapter since it can deal with a variety of issues related to having a clean print with the proper dimensional accuracy. Every single material by every single manufacturer on every single machine will have slightly different slicer settings in order to achieve the highest quality print. That being said, I go over my personal settings for each material in “Materials and Their Settings” chapter in this book, and Polymaker goes over in detail the idea behind material science in their chapter. I sincerely suggest everyone reads that “Material Science” chapter, since fully understanding it will help you to dial in your slicer settings without even reading this chapter.

When covering this chapter in my first edition, I went over the old Cura – Version 15.04.6 to be specific. Since that first book, Cura has entirely redone their interface, along with including the ability to tweak just about anything you can think of. I originally only covered Cura settings because it was the best free slicer in my opinion, but as of their continual updates, I find it better than almost any paid option as well.

Many makers prefer Simplify3D, but over the past few years I personally feel Cura offers just as many options as Simplify does, and they seem to update it much more frequently than S3D.

That said, since the last edition of my book, two free slicers have grown a lot in popularity, those being PrusaSlicer and IdeaMaker. If you do not like Cura, or you find the interface confusing, I would definitely suggest downloading those and trying them out. I will only be covering Cura in this chapter, but many of the settings will overlap with those options.

It may take a little while to get used to everything, but just about anything you want to tweak is now available. That is why this chapter is such a long and all-encompassing one.

Printers such as Zortrax, MakerBot, and many others will require their own proprietary slicer without these options, but you really should not be experiencing any settings issues on those machines when using their proprietary material. Many other printers such as Qidi and FlashForge also have their own slicers, but you can get Cura to work with them with some minor tweaks when setting up a new printer. If you have a particular machine you want to use with Cura, then you can search online to see how to properly set it up.

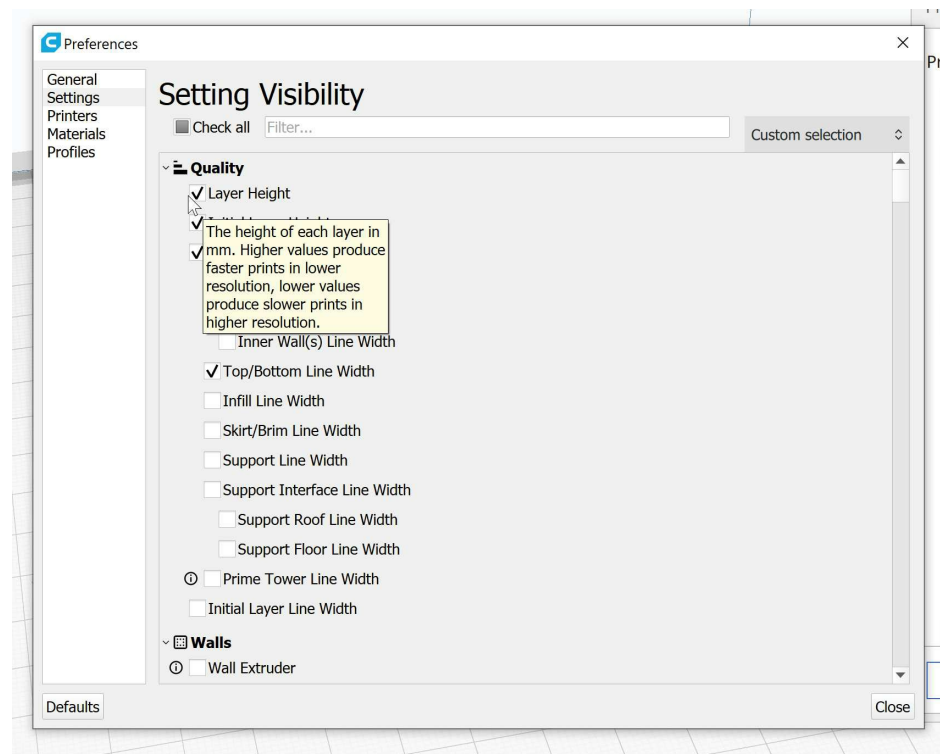
I will specifically be using Cura 4.11.0, but they are frequently releasing updates with increased options, so it is difficult for me to say what may have

changed by the time you read this book.

The following examples will be with printing in PLA on a direct extruder with a gear ratio, though you can get more specific settings in the “Materials and Their Settings” chapter. If you would like some further detail you can always visit my 3D Print General YouTube channel which goes more in-depth on a couple of these options. I now even have a two part series titled “Cura Slicer Settings Part 1/2 - 3D Printing 104”, with the part two being titled “2/2”. These two videos combined are over an hour long and I cover just about every single thing I can think of. You will definitely benefit from watching those.

About Using Cura 4.0 and Newer

In order to see expanded options, make sure you are in the “Custom” print setup, and not “Recommended”. You can save profiles for specific materials and qualities, but we will just be going over what each option does.



You can then click on the icon next to any section to see the expanded settings. When in the custom selection you can choose exactly what options you want to tweak. For anything you are not sure what it does, you can always scroll over it to be given a definition. I will personally only be going over the options I tweak, because there are literally hundreds of settings available.

Quality

Quality			
Layer Height		0.2	mm
Initial Layer Height		0.3	mm
Line Width		0.4	mm
Top/Bottom Line Width	 	0.26	mm

All of the factors I will be going over in this Quality section are the same for both Bowden and direct extruders.

The layer height of your print is very dependent on the nozzle size of your printer as well as the quality of the print you will like to achieve. A print at 0.1mm layer heights will take three times as long as one at 0.3mm layer heights, with the same nozzle diameter and printer speeds, since it will have 3 times the amount of layers. You can get up to roughly 75% your nozzle diameter in layer height with reliable results, but remember that you sacrifice quality for time the higher you go with the layer heights. You can also print as low of layer heights as roughly 25% the nozzle diameter without decreasing your quality.

Changing your layer height should not affect the amount of material you are using, just the time involved to print. The longer the print though, the higher the chance of experiencing a failure at some point. Not only will the print be longer due to the smaller layer height, you will likely have to run your extrusion speeds slower as well.

The layer height is also affected by your Z-Axis leadscrews/threaded rods. The diameter and pitch can affect the quality due to where your carriage is on its rotation. There is a handy calculator over at www.PrusaPrinters.org/calculator if you don't want to do any math. You just choose your motor step angle (labeled on your Z axis motors), your desired layer height, and your leadscrew pitch. Most of my machines are 1.8° step angle and a leadscrew pitch of 2mm/revolution on an M8 leadscrew. This means I can tweak my layer heights on a 0.01mm basis without any issues. But if you have an M5 threaded rod with a pitch of 0.8mm/revolution, you will have to tweak on a 0.014mm basis for best results. This means instead of 0.25mm layer heights, you should actually go for 0.248mm or 0.252mm.

This is a bit technical and will definitely lead to cleaner results, but I have printed outside of these suggested ranges in the past without much of a difference. This is for when you want to make sure everything is as tuned in

as possible. This will make the biggest difference when using inexpensive machines. Many users prefer to print at 0.16mm layer heights, but my go to is 0.2mm.

The Initial Layer Height is actually for bed adhesion more so than quality. This allows you to have a thicker first layer in order to make sure material sticks properly to the build plate. This number should always be at least as thick as your normal layer height, and should only be increased up to 75% of the nozzle diameter. I always take advantage of this because getting the first layer to stick is easiest with a thick first layer.

This is a big reason dealing with small nozzles is so difficult to get the first layer to stick properly. I have used a 0.15mm diameter nozzle before and it took me over a half hour of restarting the print to get that first layer Z-height distance correct. This is because the thickest I could print my initial layer height at was 0.11mm after factoring in the RepRap Calculator limitations. You can imagine that you have to be much more precise with your Z-height on that first layer when printing at a 0.11mm layer height vs 0.3mm, since any build plate deviations will be more noticeable.

The Line Width is usually just your nozzle diameter. Many makers suggest slightly tweaking this, so you can play around with what others suggest, but I personally always use the nozzle diameter for the line width I want to achieve. If you want thin 0.25mm line widths, I would highly recommend using a 0.25mm diameter nozzle instead of just tweaking this for a 0.4mm nozzle. I have actually been playing around with making my line width 10% larger than my nozzle diameter and been having good results, but I do not have that in my settings pictured above. If you would like to go the route of increasing your line width from your nozzle diameter by 10%, a 0.4mm nozzle would have a 0.44mm line width. You can watch other makers go outside this 10% range, just do so knowing that it might be smarter to just change your actual nozzle diameter.

The next setting is something I only started recently using, and that is the Top/Bottom Line Width. I had always kept this at the same as my standard line width, but on some models that may result in gaps on the top of your print. This is covered more in the “Gaps on Top Layers” chapter, but essentially if you reduce your top line width to as low as possible with your nozzle diameter, you are less likely to have gaps on the top of your print. Read that chapter for a further explanation.

There are roughly a dozen other settings you can tweak in Cura under Quality, but I personally do not change any of them, since they all deal with changing the line width of specific sections of the print. There may be

particular applications in which you want to do this, but I always stick to the line width of the nozzle diameter I am using - except for the top layer.

The 3D printing community is always learning and growing when it comes to how you can change your slicer settings to improve your print, so playing around is encouraged. Whenever I see improved results, I make sure to factor them into my future prints.

Walls

 Walls 	
Wall Thickness	0.8 mm
<i>Wall Line Count</i>	f_x 2
Optimize Wall Printing Order	☒
Outer Before Inner Walls	☐
Fill Gaps Between Walls	Everywhere 
Filter Out Tiny Gaps	☐
<i>Print Thin Walls</i>	 ☒
<i>Z Seam Alignment</i>	Sharpest Corner 

In the last edition of my book, this and the next section were part of the “Shell” section on Cura, though they recently split it up.

Within the walls section you can tweak how your outer walls are printed. The first option I have open is the Wall Thickness. The wall thickness should always be a multiple of your nozzle diameter (or rather your line width) and should be as thick as required for your individual part. You can edit this by either changing the “Wall Thickness” or “Wall Line Count”.

Unless printing something in vase mode (spiralize the outer contour in special mode), you will almost always want at least a minimum of two shells (3-4 on very small line widths). This is not only for strength, but for infill overlap to make sure the outer surface is not ugly due to the infill.

I personally print shells at 0.8mm for my 0.4mm nozzle, and 0.75mm for my 0.15mm nozzle. Anything less than this and the part will be too brittle.

If your particular part requires strength from the outer direction, such as when I print skateboard wheels, you can increase the shell thickness rather than increasing the infill percentage, since it needs strength in that direction. On the many skateboard wheels I have tested and worked with, I increased the wall thickness until it was 100% filled via shells. This is why it is important to understand which direction your part requires strength in before choosing to increase infill percentage, wall thickness, or both. For an average part I will keep this to a multiple of 2-3x the nozzle diameter, with a minimum of 0.75mm. For any part that requires strength, I normally bump this up to be at least 4 shell walls, and increase depending on the direction that the part needs its strength.

Many people prefer a minimum of 1.2mm thickness for shells, so it is up to you to determine what you think will work best here. Just do not go too thin or your part will be weak.

Optimize Wall Printing Order reduces the number of retractions and the distances travelled and will benefit most parts. It should improve speed and quality on most prints, but if you are starting to see defects going up in the Z-direction of your print or an increased slice time – turn it off.

Fill Gaps Between Walls is a great addition, since it will fill the gaps between walls where no walls fit. This is covered more in the “Gaps in Walls” chapter if you want to learn more, but generally you will want this set to “Everywhere”.

If you have a lot of tiny gaps to cover causing blobs, you can check “Filter Out Tiny Gaps” to help.

Print Thin Walls refers to the ability to print walls that are thinner than your nozzle diameter or line width. This is a neat feature since it now allows for parts previously unprintable to be printed. If you have a wall on your print that is 0.3mm thick, but your line width is 0.4mm, then previous slicers would just make it so that the wall doesn’t print at all. Do keep in mind though that this will mean those walls will not be to the proper dimensions. It is recommended you reduce your nozzle diameter and line width if you really need proper dimensions on very thin walls.

The Z Seam Alignment is where the printer will decide to go up and start the next layer. If you have ever seen a seam going up your print, it is because your printer was moving up in the Z-direction to start a new layer right at that spot, for every layer. These seams are just about impossible to avoid entirely, but you can determine where they are placed. If you are printing a part without corners, it is going to be impossible to not see seams. “Shortest” will choose the fastest printing method, which will likely lead to a seam on one part of your print, but it will definitely print a bit faster. “Random” is as it sounds, and I don’t really see any benefit to using it. I suggest sticking with “Sharpest Corner” as to hide this seam on the corners of your print.

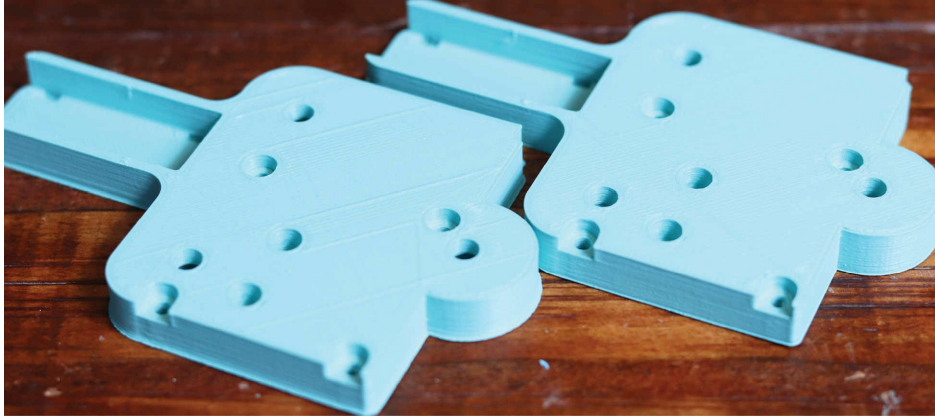
Top/Bottom

This section will cover how your top and bottom layers are printed. The Top/Bottom Thickness is how many 100% filled layers will be printed above and below the infill, and is a multiple of your layer height. This number will have to be tweaked depending on how much infill you use and how thick your model is, but I always suggest having a minimum of 1mm in top layers. This is to make sure you do not have a “pitted” or “pillowed” top to your prints. While it is important to keep the top layer thickness at a minimum of 1mm, if you are still seeing a pitted effect, it is probably better to just increase the infill percentage so less bridging is required. The bottom layers are not as important since they do not have to bridge over anything. I still will normally set this to at least 4x the layer height, with a minimum of 0.6mm being laid down for bottom layers.

Top/Bottom			
Top/Bottom Thickness	↺ f_x	1.0	mm
Top Thickness		1.0	mm
Bottom Thickness	↺ f_x	0.8	mm
Monotonic Top/Bottom Order	↺	☒	
Top/Bottom Line Directions	↺	90	
Enable Ironing		☐	

Remember that this top/bottom thickness is going to be working off a multiple of your layer height. So if you are printing 0.3mm layer heights, you will need to round to the nearest thickness in relation to that. Rather than having 0.8mm thickness for the bottom, you would want to go up to 0.9mm. Cura will round this out for you though, and you can actually choose “Top Layers” and “Bottom Layers” if you would rather do that instead of using millimeters.

Monotonic Top/Bottom Order is a very new feature to Cura and I suggest that everyone turns it on. If you get a print with bulges on the top layer, it is likely from your printer taking the quickest tool path for your top layer, rather than printing it in order. The image below shows a print on the left without checking Monotonic Top/Bottom Order, and the right is with it checked on.

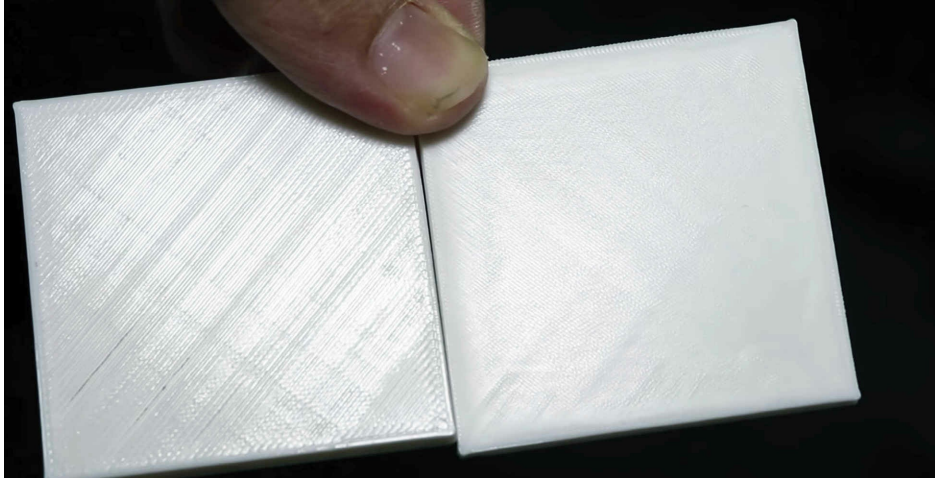


Monotonic means your printer will print lines in an ordering that causes them to always overlap with the adjacent lines in a single direction. This takes slightly more time to print, but makes flat surfaces look more consistent. This is a must for any print that has a large flat top to it.

Top/Bottom Line Directions is only something I recently started tweaking. I would suggest reading the “Gaps on Top Layers” chapter for a detailed explanation.

Ironing is a very unique setting that you definitely will not always want turned on, but does have a cool ability to iron and smooth the top of your prints. I believe Monotonic Top Layers does a good enough job, but if you need the part to be extra smooth, then you can play around with those settings.

Ironing has your nozzle go over the top surfaces an additional time, but without extruding any material in order to melt the plastic on top further, creating an extremely smooth top surface. This will only work on parts that have flat tops and is not needed on parts that are entirely curved, but if you ever wanted to remove those pesky layer lines on top surfaces, you can now do so. I have a video on the topic on my YouTube channel, but it was from a few years ago and I actually rarely use ironing now.



The photo above shows a print without ironing on the left, and with ironing on the right. This will definitely need honing in for your particular machine, as it took me a couple dozen prints to get the results you see above.

Use this setting at your own discretion and play around before printing a part you need to come out clean.

Infill

This section is what you will want to change if you need to increase the strength of your part in the top/bottom direction, reduce the “pitted” effect on your print, or decrease the time required for your part to print. Everything explained below will be the same on both a direct and Bowden extruder.

Infill		
Infill Density	↺	20.0 %
Infill Pattern	↺ f*	Grid ▾
Infill Overlap Percentage	↺	10.0 %
Infill Layer Thickness		0.2 mm
Infill Before Walls		☐

You will have to recognize how thick your part is before realizing how much infill you require. If your whole part is made up of thin walls, it is likely the infill percentage will not make any difference at all, since the majority of the print will be filled via shell walls. But if you are dealing with a large cube, it will be a deciding factor in a successful or unsuccessful print.

Most models can print successfully at 10% infill or lower, but that is not true for large/thick models. Think about the top surface contour of your print. If it is a flat top that needs to bridge a large gap, you are likely going to need to increase this infill to a minimum of 20%.

Personally, most parts that I print that do not need to be mechanically strong, I print at about 8%. So make sure you know your geometry and application since you can save a lot of time and material printing by reducing this number.

To be honest, most parts you print will not require higher infill than 50%, and you will get diminishing results when you go much higher than that. You will use a lot of material to print at 100% infill and your printer will take a lot longer to complete, when you will likely get a lower quality print without much increased strength. I will rarely print something at 75% infill and almost never at 100% (other than just for testing materials). The majority of my decorative or prototype pieces are at 8-15% infill, parts I mechanically use are 20-40% infill, and very strong parts I go around 50%. There are many parts that can actually be printed below 10%, even 0%, if the geometry calls for it and you put enough top layers in your Top/Bottom Thickness. All that said – I do print my 3D printed gun frames at 99% infill, since that is what is suggested in the guides and I do not want to mess around with what has

already proven to work.

The Infill Pattern refers to the structure shape of the infill. While you are given plenty of options, I almost always keep this on “Grid” or “Triangles”. Hexagons are one of the strongest shapes in this regard, but your printer nozzle has to go over the same line twice in order to actually make the shape. Because of this (and because Cura does not offer the Hexagonal option), I go with the second best shape being a triangle. Triangle infill can be printed extremely fast and still has a lot of strength properties.

Makers online say they get great results from some of the other options, I just stick with “Triangles” and “Grid”. “Gyroid” is a really interesting option and looks cool, but might not be really needed. Then there are 3D infill options that might help for specific use cases. When trying to print something that is bullet proof, I played around with the “Cubic Subdivision” pattern, since it allow for high density at the top and bottom, and low in the middle, which helped when I was trying to “catch” a bullet. Feel free to play around with these and see if you prefer one over the other. I also recommend watching CNC Kitchen’s video titled “TESTING 3D printed INFILL PATTERNS for their STRENGTH”, where Stefan tests out everything and gives you real results. I recommend watching most of his videos regardless since they are very informative.

Infill Overlap Percentage is the percent that your infill overlaps onto the shell walls. When this number is set too high, along with not having enough shell walls, you can get what I called a “Veiny” print in the diagnostic section. Too low of an infill overlap percentage can result in infill that rattles around detached from the walls, decreasing the strength of your part. I almost always reduce the standard set by the slicer and prefer a number around 8%-12%. This, in combination with 2-3 shell walls, will result in a strong print without any infill veins showing throw the outside. You can bump this up to around 20%, but you may start to see the infill show on the outside of your print.

Please note this is for opaque materials. If you are dealing with a translucent material you are going to need to increase the shell walls drastically if you do not want to be able to see the infill. That is not a veiny print, but rather a print you can see through.

Another way to reduce this “veiny” look is to check on the “Infill Before Walls”. This makes sure your infill prints before your shell walls, reducing the likelihood you will get a “veiny” print. That said, I prefer this unchecked for the majority of my prints.

Finally, the infill layer thickness is just what it sounds like. For well over 95% of prints I just keep this the same as the rest of my layer heights. But

let's say we are printing a large detailed print in which the strength does not really matter. We may set our layer heights to be 0.1mm, but the quality only matters on the outside walls. You can set your infill layer thickness to be 0.2mm, meaning that it only prints the infill every two layers – drastically speeding up the time required to print. This could save you hours of print time.

There are at least a dozen other options available in which I personally do not tweak in this section.

Material

Material				▼
Printing Temperature	↺ f_x	215.0	°C	
Build Plate Temperature	↻ ↺	55.0	°C	
Flow		100.0	%	

Printing Temperature is very straightforward and will be the temperature in which your hotend is set to. This is entirely dependent on the material you are using and the printing temperature will be tweaked depending on the nozzle diameter and layer height. As covered thoroughly in the “Nozzle Clogs” chapter, if you have the hotend set to the wrong temperature, you can have quite an annoying cleanup on your hands. Be sure to refer to the “Material Science” chapter to understand a bit more about melting points and what may work for different filaments and how different print speeds and layer heights may affect your extrusion temperature. Below are generic print temperature ranges for different materials.

PLA: 180°C – 220°C

ABS: 235°C – 265°C

ASA: 230°C – 255°C

PETG: 245°C – 255°C

Cheetah by Ninjabot: 223°C - 235°C

PCTPE: 232°C - 235°C

Nylon 910: 245°C – 252°C

Polycarbonate ABS: 267°C – 275°C

As mentioned elsewhere, you may need to tweak these settings depending on the manufacturer you purchase from and the machine you are using.

Remember that you will need an all-metal hotend if you want to print consistently above 240°C.

Build plate temperatures for specific materials are covered in the “Bed Adhesion” chapter, so you will find the generic temperature ranges I like to use for extruding different materials there. I personally have my build plate temperature for PLA set to 55°C now instead of 60°C in order to help reduce “Elephant Foot”.

Assuming you have your E-steps dialed in, as explained in the “Over and Under Extrusion” chapter in this book, you will not need to change the flow from 100%. But, if you wanted to change your extrusion rates on the fly, you

can change it in the flow section. 101% will extrude 1% more than what your current E-steps are set to.

This flow percentage is the section that you can change if you are unable to flash your machine or give G-code commands to set your E-steps. This isn't ideal but it should have the same results as changing your E-steps.

Speed

Speed					
Print Speed	50.0	mm/s	Outer Wall Acceleration	f_x	500.0 mm/s ²
Infill Speed	50.0	mm/s	Inner Wall Acceleration		800.0 mm/s ²
Wall Speed	25.0	mm/s	Top/Bottom Acceleration		800.0 mm/s ²
Outer Wall Speed	25.0	mm/s	Travel Acceleration		1200.0 mm/s ²
Inner Wall Speed	f_x 45.0	mm/s	Initial Layer Acceleration		800.0 mm/s ²
Top/Bottom Speed	25.0	mm/s	Enable Jerk Control		☐
Support Speed	25.0	mm/s	Print Jerk		10.0 mm/s
Travel Speed	150.0	mm/s	Infill Jerk		10.0 mm/s
Initial Layer Speed	20.0	mm/s	Wall Jerk		10.0 mm/s
Z Hop Speed	3.0	mm/s	Outer Wall Jerk	f_x	8.0 mm/s
Enable Acceleration Control	☒		Inner Wall Jerk		10.0 mm/s
Print Acceleration	800.0	mm/s ²	Top/Bottom Jerk		10.0 mm/s
Infill Acceleration	800.0	mm/s ²	Support Jerk		10.0 mm/s
Wall Acceleration	800.0	mm/s ²	Travel Jerk	f_x	15.0 mm/s

All of the numbers you see will need to be tweaked for Bowden machines, as explained in the “Materials and their Settings” chapter. But these speeds will also have to be tweaked depending on your extruder and hotend setup, and most importantly – your printer frame. CoreXY and Delta machines can definitely print faster than standard Cartesian ones, since you won’t have to worry as much about increased accelerations causing issues. If you are using a non-geared extruder, you will need to run much slower than if you have a geared one.

These are the settings that will be most tweaked depending on your machine, quality, and the material you are using. As mentioned elsewhere, if you are experiencing issues with the speed your machine is printing at, you should reduce it so that it is no faster than 100x the nozzle diameter. If you have a well-built geared extruder machine that is printing at mid-range layer heights (~50% nozzle diameter), you can actually get this number to 300x the diameter of the nozzle or higher. I would not recommend anything near those speeds if you are experiencing issues such as extruder motor skips or rattling of your machine.

Keep in mind that the amount your hotend can actually successfully extrude will also determine your maximum print speeds. An E3D Volcano hotend can push out a lot more volume per second than a standard E3D V6, meaning it can have a higher print speed with large nozzles and large layer heights.

You should definitely see the video I published entitled “How Fast Can You 3D Print?” where I cover all of these details which you should know, as well as read the “Speed Limitations” chapter.

The infill speed should almost always be the same speed as your print speed, while your outer wall should be slowed down a bit to make sure it has the best surface quality. You will also want to make sure your initial layer is

about 50% your print speeds, with it tweaked even lower if you are having difficulty getting that first layer to stick. A generally good range for your first layer speed is anywhere between 15mm/s to 25mm/s, depending on the material being used. There is no need to go fast on that first layer since it is the most likely layer to make the rest of your print fail if it does not get laid down perfectly.

Your travel speed is how fast the hotend moves when not extruding filament, and this can be bumped up fairly high, especially on Bowden machines. As long as your printer isn't rattling, you can get up to 150mm/s without much of an issue, and even higher with a light carriage. I have recently been playing around with speeds of over 200mm/s since it should reduce oozing and will not affect print quality, but keep in mind these speeds may not even be reached without a high enough acceleration.

Acceleration refers to how fast your printer gets to your print speed, and jerk refers to the initial speed your hotend will start moving at after a full stop. Your printer will start from a complete stop at your jerk speed and then accelerate to your print speed. In physics, "jerk" refers to something else, but in 3D printing it refers to the instantaneous speed from a complete stop.

You can set your acceleration controls in your printer firmware, but it can always be tweaked right here in Cura. If you have ghosting/echoing in your prints, a rattling machine, or ugly outer surfaces - reduce your acceleration and jerk. Reducing these will obviously slow your print down, but it will help immensely. Start with the numbers I show above for a direct drive machine and increase them if everything turns out fine. If not, there is likely something else that is wrong (such as a loose carriage, belts, or other issue explained in this book).

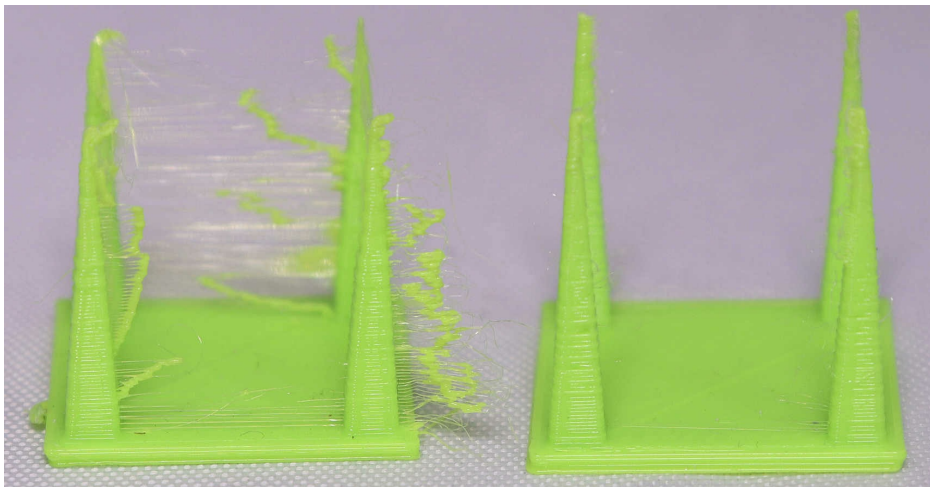
This is also where limitations for your machine come into play. You will never reach your print speed if your acceleration is not set high enough, or if there is not enough room to actually accelerate and decelerate. You can set your print speed to 1000mm/s, but without the space and high enough acceleration, it will never actually be reached. This is why small prints will not have a linear reduction in print time the more you increase your print speeds.

I personally prefer to wait a bit longer for my print to complete and have my acceleration and jerk settings very low in order to guarantee a clean and successful print. You will likely hear others printing much faster than this, but if you are using a non-delta machine that cost under \$1000, it is likely going to be hard for you to achieve those results.

Travel

Travel	
Enable Retraction	☒
Retraction Distance	1.5 mm
Retraction Speed	30.0 mm/s
Retraction Minimum Travel	0.9 mm
Combing Mode	All
Avoid Printed Parts When Traveling	☒
Avoid Supports When Traveling	☒
Z Hop When Retracted	☐

Retraction should be enabled on the vast majority of your prints and will need to be tweaked based off of the material and if you are using a Bowden or Direct extruder setup (with the numbers shown above being for PLA on a Hemera direct extruder). Bowden machines will need higher numbers than I am showing, and even standard direct extruders may need a higher number. Basically - the shorter the distance between your extruder and hotend, the lower you will want these numbers.



Having your retraction dialed in is the number 1 way to reduce the stringiness and “hairy” prints. There are a few models that may require retraction turned off, but you will almost always want it on.

The print on the previous page and on the left is with retraction turned off, and the right is with it turned on to the settings I have shown above. Please keep in mind this model is designed to torture test your machine, and you

may not need quite as high of retraction settings on every model. Going too high of retraction may result in an under extruded part or slight holes in your print.

The retraction distance refers to how far the material retreats when retracting, retraction speed refers to how fast, and minimum travel refers to the amount of travel required for any retraction to happen at all. If you have a part with extremely small pegs (for instance - a very small fence on an architectural model) and they are still coming out “hairy” after using the settings I example, you may need to decrease this minimum travel number.

Bowden machines will need these numbers bumped up, as explained in the “Materials and their Settings” chapter.

Particular materials also require different retraction settings. I have found that PETG is far stringier than PLA, requiring higher retraction settings. You can understand this further in the “Material Science” chapter.

It is very easy to clean up a minor amount of stringiness via a heat gun and a razor.

Combing refers to the printer head following the path of the print rather than attempting to clear gaps. This will help prevent “hairy” sides of a print. When combing is set to “off”, the extruder moves straight from the starting point to the end point and will always retract. Most of the time I will leave combing set to “all”, but there are specific times when I need combing turned off.

When I work with very large layer heights – anything 0.4mm or higher, I will turn combing off. This is because the layers are so thick that when the printer travels from one point to another it will run into the infill. Even when I have “avoid printed parts when travelling” checked, the printer seems to want to run into the infill lines. You won’t really notice this on low layer heights.

When combing is turned off, it will retract, do a z-hop if you have it turned on, and then travel to its next position. As mentioned – this can add to your ooze and hairy issues, but sometimes it is needed. For instance, when I was using the E3D Super Volcano with a 1.4mm nozzle at 1.0mm layer heights, the print was just about impossible when combing was turned on.

Both avoid printed parts when traveling and avoid supports when travelling refers to what you think it would – the printer head steers clear of your print while moving from section to section. This will clearly add to print time considering the print head will not be taking the fastest path, but it will help avoid knocking your parts over. If you use a Z-hop, you likely won’t need these turned on, but I sincerely recommend not using a Z-hop on inexpensive printers. Z-hop should only be used on well-built machines or Delta printers. This is something I only recently learned.

Z hop when retracting refers to the amount the printer will raise the hotend or lower the print bed in the Z-direction after retracting, as to not knock over small pieces when traveling between sections. If you are constantly getting parts knocked over even though you took all of the precautions mentioned in the “Bed Adhesion” chapter, you may need to turn on a Z-hop and increase the number. The larger the nozzle diameter and layer height, the more you will likely want to increase this. Also refer to the “Parts Being Knocked Over” chapter if required. That said – if you have “Avoid Printed Parts When Travelling” turned on, this really should not be an issue. As mentioned though, I try and avoid using a Z-hop unless using a Delta or well-built machine.

If you ever hear some random loud clicks or noises during your print that you are having trouble figuring out where they are coming from, it may be from your nozzle hitting the print or support as it goes over it. This shouldn't be an issue if you are avoiding printed parts, but your slicer may be taking a strange toolpath. For the vast majority of my prints on well-built machines, I have my Z-hop set to the same as my layer height and will tweak accordingly if needed. If the clicking continues – turn combing off.

Cooling

✖ Cooling ▼		
Enable Print Cooling		☒
Fan Speed	f_x	100.0 %
Regular Fan Speed at Height		0.7 mm
Minimum Layer Time		3.0 s
Lift Head		☒

Cooling refers to when your active cooling fan will be engaged. Refer to the diagram of a 3D printer in the beginning of this book to know exactly which part I am talking about. This is crucial for getting the cleanest print possible on many different types of materials. If you are printing PLA without an active cooling fan, it is certain you are not achieving the best results you can and may have curling of layers.

This is not true though with specific types of materials, since you will reduce your layer adhesion or increase warping/delamination. You need to know what material you are working with in order to understand if you need an active cooling fan on or off. If you are working with a material that warps, you will almost certainly want it off. This will decrease the surface quality of your print and you won't be able to achieve as steep of angles without support structures.

You do not want your active cooling fan to blow on your first layer since it can hurt with your bed adhesion. This is why I set the "Fan full on at height" number to 0.5mm – 0.7mm on most prints. It is rare that a material will call for a fan speed in-between 0 and 100%, so it is normally set to 100% on most materials that require a fan.

If I am printing a large ABS part that comes to a small point, I may actually turn on the active cooling fan and turn this "Fan full on at height" number to the height in which the point starts. If you have an active cooling fan on a medium-large ABS part, or any other high-warping material, it is likely you will end up with a failed print. This is why you will want the active cooling fan turned off for the majority of ABS prints, unless the geometry calls for it.

This active cooling fan can also play into your nozzle not maintaining a specific set temperature if the fan is blowing directly on your heater block, or if you are not using a silicone sock.

Minimal layer time refers to the amount of time required before starting a new layer. If a layer completes its tool path faster than this amount of time,

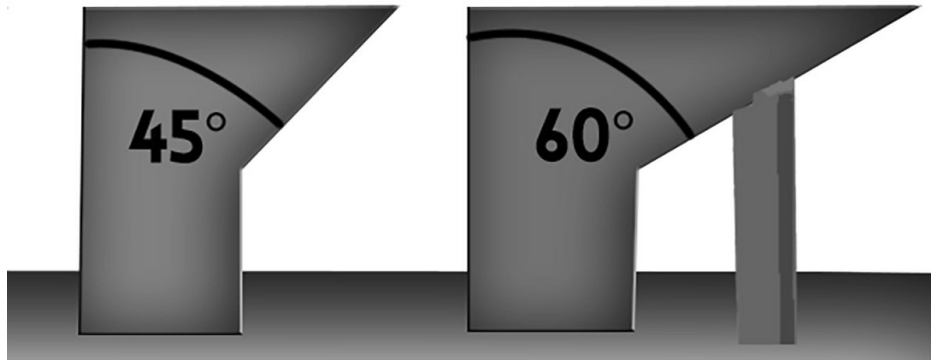
your printer will pause until the time has passed before starting the next layer. This setting should almost always be accompanied with a “Lift Head” that you see right below it.

This “Lift Head” does just as you think it would. If a layer finishes faster than your minimal layer time, it will lift your hotend and pause until the correct amount of time has passed. For the example photo above, if a layer takes two seconds to complete, your nozzle will lift and remain there for one second before starting the next layer. This can add to the stringiness of your part but will make sure your top section is not a melted mess.

Most of the time you will not require this number to be higher than 5, since 5 seconds is long enough for the vast majority of materials to properly cool. If set to 1 second or lower, you will get very ugly, melted pointed tops or curling of layers. Any model that doesn't come to a single point will not even be affected by this minimum layer time, unless set very high. If you are just printing one skinny part, you will notice that it may come out hairy due to constant head lifting from the minimum layer time not being hit. In these instances, you can print 2 or 3 of the model without increasing the print time at all, and may actually result in a cleaner print as well.

Support

The overhang angle for support refers to the minimum angle required before support material is laid down, vertical being 0°. This may seem a bit confusing so it is best described by the image below:



The general rule of thumb is support material will be needed on overhangs of 45° or greater when not using an active cooling fan. PLA can actually cleanly lay down angles of a higher degree if everything is set up properly and you are running an active cooling fan. There are models you can find on www.Thingiverse.com that allow you to test the highest angle your printer and material can achieve without supports. Just search for “overhang test”.

Support			
Generate Support	☒		
Support Placement	☒	Everywhere	Support X/Y Distance
Support Overhang Angle	☒	55.0°	Enable Support Interface
Support Pattern	☒	Lines	Support Interface Thickness
Support Density	☒	12.0%	Support Roof Thickness
Enable Support Brim	☒		Support Floor Thickness
Support Z Distance	☒	0.2 mm	Support Interface Resolution
Support Top Distance	☒	0.2 mm	Support Interface Density
Support Bottom Distance	☒	0.2 mm	Support Interface Pattern

Materials may actually be able to achieve different overhangs without the need for support depending on your other slicer settings, particularly your layer heights. This is covered a bit in the “Materials Science” chapter, as well as a video on my YouTube channel titled “How to Avoid Needing Support Material”.

For ABS, I have this number set to 45°, and I will hone it in for any angles I notice scarring. In general, for most materials I will not go lower than 40° or higher than 60°. If you are printing without an active cooling fan, you will not be able to achieve as steep of angles, which is why I have ABS set lower than PLA. For most PLA prints I have my support angle set to 55°.

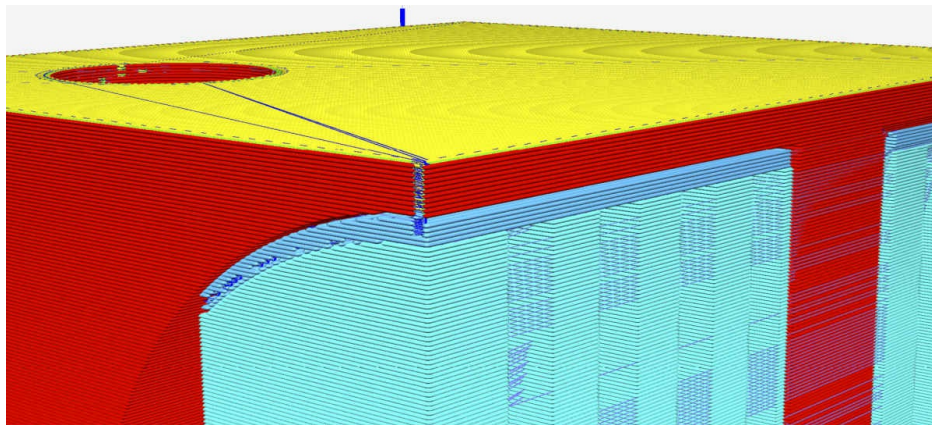
You will usually want your support placement to be “Everywhere”, unless the model you are printing was designed to only require “Touching

Buildplate”. “Touching Buildplate” is what you assume it would be, it will only place support structures where they can easily touch the build plate. This is often not desired on most prints. I will use “Touching Buildplate” if there are angles that require support touching the build plate, but there are a lot of holes inside the print that do not require support structures.

The support pattern I almost always have set to Zig Zag, though you can use Lines to save filament, or Grid to increase strength of the supports. When using the Support Interface explained shortly, your support pattern won’t make much of a difference at all. The support density should be tweaked as well if you are not using the support interface option.

Please refer to the “Experimental” section near the end of this chapter to find out how to reduce your support material even further.

Essentially, the support interface generates a dense interface between the model and the support, creating a skin at the top of the supports and below the model. This essentially works similar to a “Raft”, but for all undersides and angles. In the first edition of my book this was not available yet, so I went over my support settings just using the normal support lines.



As you can see in the image above, there is this support interface in-between the normal support and the model. The image may be hard to see on your printed book, so don’t forget to reach out to me at Sean@3DPrintGeneral.com for high definition color photos.

This means that the support pattern and support density will not affect the underside quality of your print when having support interface turned on, so I will reduce my support infill to as low as I can get it. Too low and your support may get knocked over during the print, which is why I like to keep mine at 10-12%.

I use this support interface on almost all hard materials, but if you are using flexible filaments or materials with very high layer adhesion, you will likely want this turned off. Removing parent support material can be quite difficult

on many unique materials.

The numbers you see are what works for me on PLA for both a direct drive and Bowden extruder, and may need to be tweaked further for your machine.

Your Support Z Distance is likely the most important section of your support settings, since it refers to how much gap there is between the top of your support and the underside of your print. This gap is what makes you able to remove your support structures, and anything too close is why some supports are extremely difficult to remove.

The Z distance for your support settings work on a multiple of your layer height, and slicers will round up for any number you put to the next nearest layer height. This means that if you are printing with 0.2mm layer heights, your Support Z Distance would need to either be 0.2mm, 0.4mm, 0.6mm, etc. If you were to set your Support Z Distance to 0.15mm, your slicer would round that up to being one layer gap, or 0.2mm. This is why your support structures may be easier or harder to remove depending on your layer height. Printing at 0.3mm layer heights requires at least a 0.3mm gap.

If the underside of your print is still ugly after using my suggested starting points, either increase the support interface density and/or decrease the Z distance for support. If it is proving very difficult to remove this support material based off of my suggested settings, you should increase the Z distance for support. The support X/Y distance of 0.8mm seems to work just fine on almost all models and materials, and any closer the support material may be difficult to remove. You can increase the X/Y distance if you are printing something with a lot of small pits on the side that don't require support, but in general I keep this set to 0.8mm.

I created a video titled "Detailed Cura Support Settings" where I take a look at every aspect of this and show how it makes a difference in your prints. I spent a long time making this video covering everything, so I really suggest checking it out if you are having a lot of support difficulties.

Build Plate Adhesion

 Build Plate Adhesion 			
Build Plate Adhesion Type	 	Brim 	
Brim Width		0.0	mm
Brim Line Count	  	15	
Brim Only on Outside		☒	

Having a skirt will allow for the material to purge a bit at the beginning of your print and for you to double check that the bed is level, but will not touch your model. You will only need one or two lines, and you will want the start distance far enough from your print so that it does not interfere with it. A distance of 3mm works fine.

For roughly 90% of my PLA prints I only use a skirt, since there is no real problems with warping. But if you ever need a part to have enhanced plate adhesion, change this to a Brim.

If you are having difficulty getting your part to stick to the bed, or if you are printing with a high warping material, you will want a Brim, as mentioned in the “Bed Adhesion” chapter. If you choose “Brim”, it will add the number of lines you choose touching the perimeter of your print. These lines are as thick as the line width you are using (which I normally keep as the nozzle diameter). So if you choose 15 lines and are using a 0.4mm nozzle, you will be adding 6mm of brim around your print. Thinner nozzle diameters will require more lines to equal that same surface area of brim.

This brim acts as an anchor your print to prevent warping and to help with bed adhesion. This brim is then removed post print. I have “Brim Only on Outside” checked for most prints, because it will prevent extra brim from being laid on holes on the inside of your print, and only be printed on the outside perimeter of your model to act as an anchor.

Keep in mind that a brim can be difficult to remove cleanly from some materials, one being PLA, which is just another reason I keep my settings to only be a skirt for the majority of PLA prints. A brim on ABS is much easier to remove than one on PLA.

Finally, your last option for platform adhesion, is to add a raft.

 Build Plate Adhesion ▼		
<i>Build Plate Adhesion Type</i>	 	Raft ▼
<i>Raft Extra Margin</i>	 	5.0 mm
Raft Air Gap		0.3 mm
Raft Top Layers		2
Raft Top Layer Thickness		0.2 mm

I honestly do not use this option often, but there are specific times I will. Many printers, such as the ones made by Zortrax, will leave an easy to remove raft on all prints standard. The Zortrax does it because the build plate is perforated and would leave an ugly underside to your print if it didn't. Other printers make this standard because they do not have a heated build plate. I will also personally use a raft when I am battling with “Elephant Foot”, as described in that chapter.

If you are printing a part that only has a lot of very small posts touching the build plate, you may want to consider adding a raft. Material has a lot easier time sticking to itself when printing than it does sticking to the build plate on its initial layer. By adding a raft with specific settings, you can make sure the bottom layers are nice and thick with plenty of adhesion, and that your print sticks to the top of it. You can then easily pop the parts off without having to battle with cleaning off a brim for a few hours. This is why a raft may be beneficial for a build plate with a lot of small parts.

These rafts are then removed after printing, but can damage the underside of your part if too close, and can leave ugly scarring if your part is too far. The “Raft Air Gap” is similar to the “Z Distance” on support structures. You can increase this raft air gap if the prints are stuck to the raft, and you can decrease it if they pop off too easily and leave an ugly underside.

Special Modes

★ Special Modes		
Mold		☐
<i>Spiralize Outer Contour</i>	 	☒
Smooth Spiralized Contours		☒

There are a few “special” modes offered by Cura, but I will only use one on rare occasion: Spiralize Outer Contour. Please make sure these two boxes are NOT checked for a standard print. I am only checking them in the example above so you can see the two options involved.

Spiralize the Outer Contour is basically “Vase Mode” for Cura. This will make it so only the outer wall and bottom layers of your part are printed. No inner walls, no infill, no top layers. This option should leave no seams on your print.

You will only want to use this when you are printing a vase or something that needs similar properties. I have a Christmas tree that I printed large in this mode that looks great, along with a couple of vases given to my mother. I also use this mode when I need to print something transparent in PolySmooth, as covered in my “Transparent 3D Prints” videos. Other than rare use cases like this, it is likely you will want to keep this option unchecked.

Mold I pretty much never use, but it will help to create a negative of your print if you wish to create a mold of it.

Experimental

Experimental		
Infill Travel Optimization		☒
Enable Draft Shield		☐
Enable Coasting		☒
Coasting Volume		0.064 mm ³
Minimum Volume Before Coasting		0.8 mm ³
Enable Conical Support		☒
Conical Support Angle		45.0 °
Conical Support Minimum Width		10.0 mm
Enable Bridge Settings		☒

This section is constantly changing and is where “Ironing” once appeared a while back. I attempt to go over some of these on my 3D Print General YouTube channel, but Cura is always being tweaked. I currently have 4 videos on experimental Cura settings, so refer to those for further information.

I have tried a few of these but have found the most useful to be Coasting and Conical Support. Coasting is so useful that I expected it to be moved out of the experimental section by now. I do not have coasting turned on for direct extruder prints, but almost always have it on for Bowden machines, especially when printing PETG.

Coasting replaces the last path of an extrusion path with an oozed travel path. The oozed material is used to print the last piece of the extrusion path in order to reduce stringing. If you are experiencing “hairy” or stringy prints regardless of tweaking your retraction settings, you will definitely want to try this out. In fact, I have this turned on for almost every Bowden print that I do, as well as for some flexible materials. The volume should just be your nozzle diameter cubed, and the minimum volume before coasting will make it so coasting is turned off for small areas. This is important to get to 0.8mm or above to prevent small parts from looking under extruded. If that is happening to you, go ahead and increase this minimum volume before coasting. Using coasting when it is not needed can result in a part that has holes on the side of it, as covered in the “Missing Layers and Holes in Prints” chapter.

“Conical Supports” is pretty awesome, as it allows the amount of support material touching your build plate to be less, and then grow at an angle until it is the size required for the underside of your print. If you go too low you can be battling with too small of a surface area touching the build plate, but the settings I have above work great for majority of prints. This can actually end up saving you a lot of material and time on large builds. Please refer to my videos titled “Cura Tricks for 3D Printing” and “Cura Experimental – Draft Shields, Support in Chunks, Conical Supports, and Printable Overhangs” where I go over this specific setting further.

A “Draft Shield” builds a wall around your print in order to help trap air and reduce draft air from getting to your print. This should, in theory, reduce the amount of warping and delamination you experience - I just personally have not noticed much of a difference. It may be worth you playing around with if you are having warping issues.

“Infill Travel Optimization” should help to reduce print times, so I actually keep it on frequently, though it doesn’t do much from my testing.

Bridge settings help to bridge sections that do not have support structures. I do not always use it, but it can be beneficial if you need to bridge large gaps.

There are many other settings here, so always keep an eye out for future videos on my channel where I test some of them out.

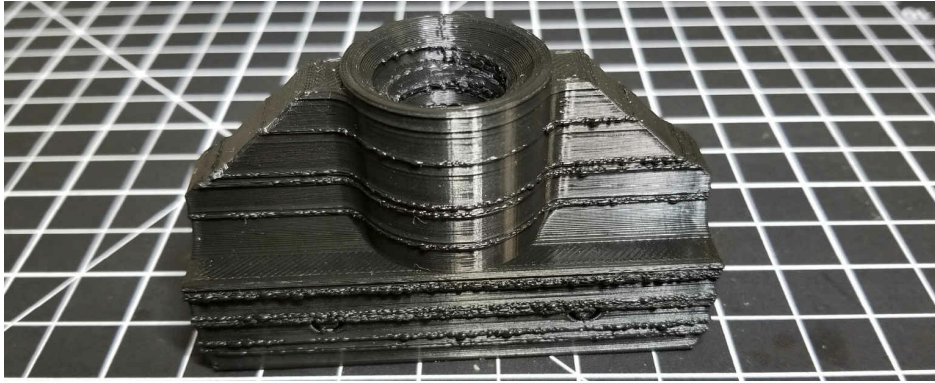
Settings on other slicing programs

Other slicing software may have these features in different sections, may call them something different, or they may not have them at all. For years Simplify 3D was the way to go if you wanted full control of your slicer settings, but as of the last couple of years Cura offers just about everything Simplify 3D does – free of cost. There are so many options available that I have well over half of them turned off. You can spend a couple of weeks just doing test prints to figure out what each one does, allowing for full customization.

I am definitely not paid by Cura/Ultimaker and have had zero contact with them other than really enjoying their updates to the slicing software. You should also check out PrusaSlicer and IdeaMaker for other options.

Squished Layers

As with the previous two editions of my book, many of the new additions have come from readers just like you. This particular failure has come from two individuals who bought my 2020 edition who emailed me for help.



(Photo Courtesy of reader John Fishman)

Check the mounts and couplers on your frame

While both individuals had different parts of their printer malfunctioning, they were both similar in nature. John, whose photo is above, had his Z leadscrew not properly attached to the stepper motor coupler. As the extruder moved up in the Z direction, the threaded leadscrew would detach from the stepper motor, and not turn again until it fell back into place.

That ugly print was fixed with something as simple as pushing that threaded leadscrew back into the coupler, and tightening the grub screw that holds it in place.

The second individual's issue was a little more confusing, but basically the way his x-axis was connected to his z-axis was loose. This means that the x axis bar could drop or move up with a bit of free play. Tightening that up fixed his issue entirely.

This leads me to believe that the majority of these issues are mechanical in nature, and if you are experiencing this you should do a full maintenance diagnosis on your printer. Make sure every part is connected to every other part as it should be, with all screws and nuts tight.

Extruder Malfunctioning

When both of these individuals emailed me, my first thought was that the extruder was malfunctioning by over-extruding and showing itself in specific smashed areas. While in neither case was this the issue, if you tried the above maintenance and are still having this issue, then my next guess would be extruder related.

Read the “Over and Under Extrusion” chapter to make sure your e-steps are correct, and double check your flow% is set to 100% on your slicer.

Take your extruder apart and make sure everything is connected as it should be without any blockages and that the filament feeds properly.

Then, if still persisting, I would even go as far as to try out a new extruder stepper motor or stepper drivers.

After only seeing this issue twice though, I would assume that this is particular to frames that are not connected together properly or have just become loose over time.

Summary of Fixes and Precautions

- Make sure your z leadscrew/threaded rod is secure in the coupler that attaches it to the stepper motor.
- Make sure your X axis is properly secured to your Z axis frame without any free play.
- Check all other mounts and make sure everything is secure.
- Disassemble extruder and make sure everything is turning as it should be.

Stepper Motors Overheating or Malfunctioning

This is a fairly broad printing failure but is essentially when one of your motors is not turning properly or is overheating. Many of these issues are covered in the “Layer Shifts” chapter in this book as well.

Stepper motors overheating

A stepper motor can be running too hot for multiple reasons. If you have an enclosed printer and are running a long ABS print, not only will the stepper motors be getting hot from standard usage, they will be trapped with ambient air of 35°C – 50°C (or even higher). Even if a print is being completed without any issues, you do not want to run your motors too hot so that they can remain as free of maintenance as possible.

Industrial grade stepper motors have magnetic cores which begin to degrade when they reach temperatures above 80°C. During warm days or prints on an enclosed machine, the stock stepper motor surface temperature will hover between 70°C and 75°C for long print durations at moderate extrusion speeds.

Along with reducing the current to the stepper driver as mentioned in next section, you can provide some external ways to cool these motors.

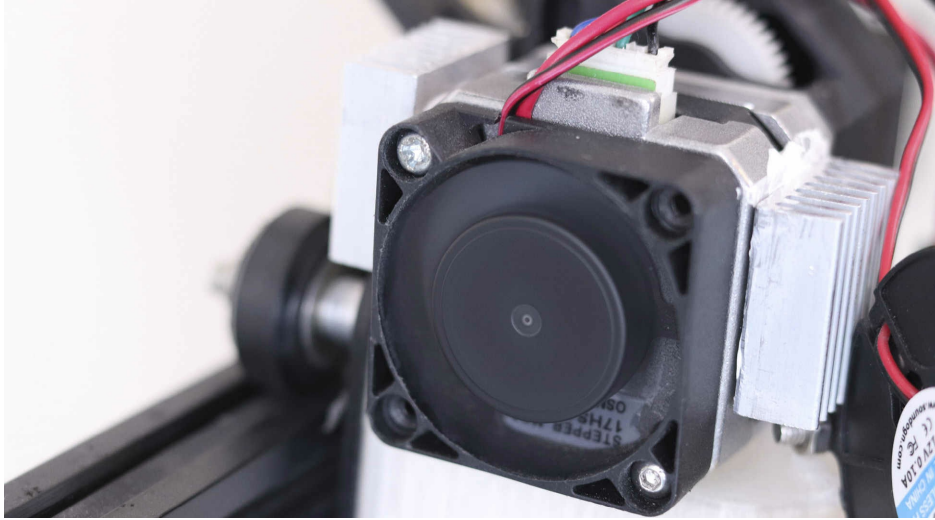
The extruder motor is one of the most common to overheat, so it can only be beneficial to add a passive heat sink. Increasing extrusion rates while maintaining a constant output temperature will require additional torque from the extrusion motor. This additional torque will require more power, which creates thermal dissipation through electro-mechanical inefficiencies – meaning this motor will become hotter the faster it runs.

After the extruder motor, the X and Y steppers are the second most likely to overheat, since the Z will only move intermittently.

A basic fix for a hot stepper motor is that you can screw a heat sink into your motor. You could also get thermal glue or some use some strong thermal double sided tape to stick the heat sink right on the stepper motor. These will increase the heat dissipation of the individual motor by expanding the area available for convective heat transfer.

I suggest to everyone that they at least put a heat sink on their extruder motor. You can touch your extruder stepper mid print and see just how hot it is getting.

Another option to keep the motor cool is that you can connect a fan that will actively blow on that stepper motor. You can wire this fan directly to the power supply on your board so that it turns on when your printer is on, allowing for cooling of any overheating stepper motor. This is not needed on most machines, but if you are running hot, this is a great way to help remedy.



If your stepper motors persist in their overheating, you may want to skip down to the section “Check Driver Current” later in this chapter.

Stepper drivers overheating

Along with your stepper itself overheating, the stepper drivers can overheat. Most drivers will come with very small heat sinks that definitely help, and if yours does not, I highly recommend applying them. Though this helps, it is not enough in itself to entirely prevent stepper drivers from overheating.

As mentioned elsewhere in this book, you will always want 1-3 active cooling fans blowing onto your board. This can do wonders when it comes to overheating drivers and other parts on your board. All pre-built machines should have these fans standard, but they can burn out or be damaged over time. If you are building your own machine, you need to include these fans.

Just remember that if you do not have a filter on these fans you will see dust accumulate which can cause the fans to fail over time. Always make sure your printer is off when it is not being used to not collect unnecessary dust, and to clean these fans and your board periodically. Almost every pre-built printer comes with these fans on the board standard, so periodically check to make sure they are spinning well.

If still experiencing overheating of your stepper drivers, you will want to check that the current going to that driver isn't over the rated limit via the methods described in the "Extruder Motor Skipping" chapter, or in the "Layer Shifts" chapter. These stepper drivers are inexpensive, so if you are continuing to have a driver overheat, it might be smart to try and replace it. Just be careful – an overheated stepper driver will be so hot to the touch it can hurt you.

Some printer boards, such as the one on the Ender 3 V2, have integrated stepper drivers, so they wouldn't be something you could change or swap out. That said, you can still change the VREF as needed, which is covered in the next section.

Check driver current

Having too low of power going to your stepper motor can cause stepper motor skips, leading to layer shifts or under extruded parts. Having your voltage set too high can lead to overheating stepper motors. This explanation is going to be the same that is covered in the “Layer Shifts” chapter.

This should not be an issue with factory made machines, but will be more common on the inexpensive DIY ones. That said, I have had to change the stepper driver on one machine that was experiencing this overheating issue, straight from the factory. In fact, there are quite a few stories I hear of VREF settings being incorrect on factory made Creality machines, so it is definitely worth checking to see if your stepper motor has the proper amount of current.

You first want to have your machine turned off and disconnect the stepper motor cable from the board. You then need to look up both the stepper motor and drivers you are using, or you can just search online for VREF values on your particular printer.

Current limits are determined in the motor and driver data sheet. You will not want to run higher than either the driver continuous current limit, or the motor current rating limit - so it is often good to have a driver that has a higher continuous current rating. I suggest going off of the continuous current limit of the motor.

Once you know what current limit you want, you then need to find out the calculation for your stepper driver to determine a VREF. You can go to the current limiting section of your stepper driver data sheet in order to figure out the VREF you will want. Current limit will equal $VREF \times 2.5$ for standard A4988, and $VREF \times 2$ for DRV8825. There is then another option by TMC where the calculation is slightly different, where $VREF = (\text{Motor current} \times 2.5) / 1.77$. To find out your ideal VREF, you can try out a handy calculator that someone made online which you can search for by typing “Stepper Driver VREF Calculator”, where you just enter the rated current of your motor and it will read out the ideal VREF numbers for each driver type.

That said – it might be difficult for you to figure out exactly what your machine needs, so it might be smart to first search online for your specific printer and its recommended VREF or voltage. This helps a lot since many people have already figured out proper settings for popular machines. Going off of TH3D’s website, a stock Ender 3 V2 should have a VREF in Volts of these numbers:

X: 1.1 – 1.2V

Y: 1.0 – 1.1V

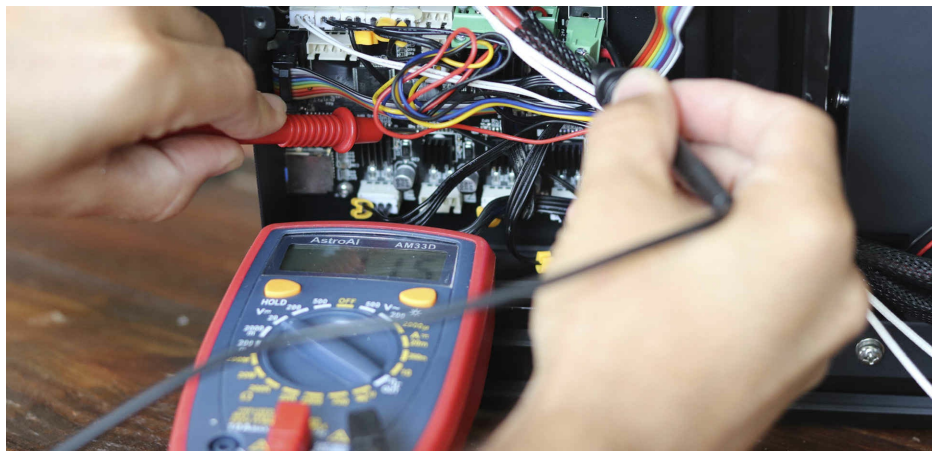
Z: 1.15 – 1.2V

E: 1.35 – 1.4V

I understand this is very confusing for someone who is new to this, just as it was for me. Essentially the VREF is the power that is being sent to your motors and it can be tweaked depending on your motor and stepper driver setup. This means that if you set your extruder motor VREF on the Ender 3 V2 to 1V or less, you are likely to experience extruder motor skips, since not enough power is being sent to the motor. If you were to set it to 1.5V or higher, you would be more likely to have your extruder motor overheat.

When dealing with the Ender 3V2 as an example, you would want your multimeter to read out a number in-between the ones provided above for a proper VREF. You can adjust accordingly from these values if your stepper motor is running too hot or having too many skips, since I do notice having a VREF of 1.15V on my Ender 3 V2 has the Z stepper motor running a bit hot. This means I set my Z-stepper motor on my Ender 3 V2 to be 1.1V.

To test this out, you will actually need to plug back in the power and turn the machine on, all while you still have access to the board. Be careful now that everything is on. You would then grab your multimeter. No real way to do this without a multimeter.



Set your multimeter to 20V DC, and touch your black negative lead to a ground pin on the stepper driver (titled GND). If you are unaware which the ground pin is, you can also touch the black negative lead to the negative section of your power supply. On my Ender 3 V2, I just touch the metal on the screw terminal where the black cable is going from my board to my power supply. Make sure to only touch a ground section with your black negative lead.

You can then clip the positive lead of your meter to the metal shaft on the screwdriver to help read everything out while you change it. If you do not

have a clip to connect the red lead to the screwdriver, you will need to test, tweak, and test again. You then touch the positive red lead (or the screwdriver if you have it clipped) to the potentiometer on the stepper driver. This is a very tiny screw like object on the driver. You will see a voltage number read out on your multimeter. Remember you need your printer powered on, so make sure you only touch a ground with your negative lead and your potentiometer with your positive lead. This number your multimeter reads out is your VREF, which you want to make sure equals the proper rating for your machine.

You can turn the screwdriver clockwise on the potentiometer to increase the voltage, and counterclockwise to decrease it (which is actually the opposite on some drivers, so just make sure you are testing after each small turn). A 1/8 turn of the potentiometer will make a drastic difference in your VREF – so make sure to not turn too fast. Turn a very small amount, retest, and then continue until your multimeter is reading the correct voltage.

Once you get the proper readout for your printer, plug everything back in and run another test print. If your VREF was originally set too high, your stepper motors should run much cooler now. If you were getting skips and layer shifts and your VREF was set too low, that problem should hopefully be fixed now after increasing the VREF.

Sometimes you will have to reduce the VREF on your extruder stepper motor if you are swapping from a full sized stepper to a pancake one, such as going from a standard Ender 3V2 extruder to something like the BIQU H2, which uses a smaller stepper motor to turn the extruder. A good rule of thumb is if you are swapping to a smaller stepper motor, you would want a lower VREF, and same is true vice versa. That said – it is always smart to check data sheets first so you know the exact range you should be in.

Unlevelled X carriage

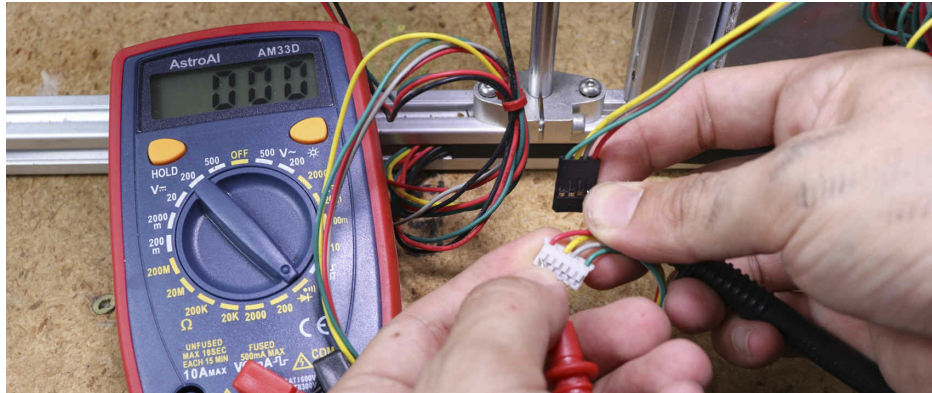
In the “Unlevelled Build Plate” chapter in this book I mention that it is important to level the right and left side of your X carriage in relation to how close it is to the build plate. These can become unlevelled over time, but you will see some massive differences if one Z stepper driver is malfunctioning. This will only be the case on printers with dual leadscrews, and not an issue on a single leadscrew printer such as a stock Ender 3V2.



If you notice your printer is looking like the photo above, you likely have a malfunctioning stepper (or stepper driver) for one of the Z-axis motors. It could also mean one of the parts on your frame is broken, so inspect closely. This, as well as any other stepper malfunctioning, will have to be remedied as follows:

Check the continuity and wiring of the stepper in question

With the frequent rattling and movement of your machine, wires can easily get caught mid print and become either disconnected or frayed to the point they are not providing continuity. The first thing you will want to do when you see a stepper is malfunctioning is to check the continuity of each wire.



If not visually frayed, you will need to take your multimeter and switch it to the continuity tab. Turn off and unplug your machine. Then unplug the connector for the stepper in question and its connector to the board.

Take either lead and touch it to one of your wires inside the connector to your motor (be sure it is touching metal). Take the other lead and repeat the process for the connector that is going to the board for the same color wire you are testing on the motor side.

If you hear a beep, that means there is no break in that wire – there is continuity from the starting point of the wire to where it connects to the motor.

Continue the same process for each colored wire. If you notice that one of the wires is not resulting in the multimeter beeping, that means there is a break in the wire somewhere. Follow that cord from the motor to the board and see if there is anything you can see with the naked eye (frayed, cut, or burnt out sections of the wire). If it easy to spot, cut that section out, re-solder to two wires, and confirm that there is now continuity.

If there is no physical damage you can spot with your naked eye, you will want to replace that wire entirely. Once you confirm all wires have continuity from the board to your motor, everything should be working properly again. If not, move on to the next step.

Plug in a different stepper motor (or plug the current stepper motor into a different driver)

You can test to see if it is your stepper or driver that is malfunctioning by swapping the stepper motors. Unplug the stepper in question and take those same wires and plug them into a different stepper you know is working properly (either on your machine or a spare that you have on hand). Attempt to move the stepper again by moving the axis in question and see if it is working properly. If it is, this means you will have to replace the motor that was malfunctioning.

If when you plug in this different stepper that you know is working, and it does not spin when moving the axis in question, you have a problem with either your stepper driver or your board. This is assuming you have confirmed that all wires in question have continuity (as described above).

You can achieve this same outcome by using the motor in question but plugging it into a different driver that you know is working.

Replace a motor if it is the problem, and move on to the next step if it is working fine.

Test your stepper driver by swapping it

Once you have confirmed that your wires all have continuity and that the stepper motor itself is working, you will want to test the stepper driver in question. This is simple and can be done with any spare stepper driver you may have, or by just testing a driver for a different motor you know is working. Remember though that this wouldn't be possible on the Ender 3V2 board, since the stepper drivers are integrated. For that type of board, you would not be able to just swap a new driver and retest.

Unplug the driver in question and plug your spare/new driver in if possible. If everything works just fine, you know it was the driver that was malfunctioning. Always make sure you are using the proper stepper motor and driver for your machine setup.

You will want to make sure that this new stepper driver doesn't overheat and get burnt out again. If you are using a different brand driver, you may need to figure out the proper VREF, as explained earlier in this chapter. If you have the proper fans blowing, and your new stepper driver over heats as well, you may unfortunately need a new board.

Final fix – replace your board

Finally – if all wires are showing continuity, the motor is proven to be working fine, the stepper driver is functioning just fine - it's likely that you unfortunately have a malfunctioning board. Boards have definitely come down in price, so research the one used on your machine.

Once you have made it this far in your testing, and you made sure you are using all the proper parts, I can almost guarantee the issue remains in the board itself. Boards that require NANO fuses will not work at all if the NANO fuse is blown, so if you are able to heat your extruder and move different axes, then it is not a NANO fuse problem.

Buy a replacement board, or get a free replacement if under warranty, and embark in the annoying process involved in unplugging and re-plugging everything (with your printer off and unplugged of course). Some machines can be extra frustrating to do this on because of the limited space you have. You unfortunately will not be able to just test the one motor in question until everything is wired and your board is flashed with the appropriate firmware.

I suggest you take a picture of your old board while wired so you can refer to it if you get lost connecting your new board.

Once everything is plugged in, and the correct firmware is flashed, test everything out. If all the proper tests above were taken, everything should now be working fine.

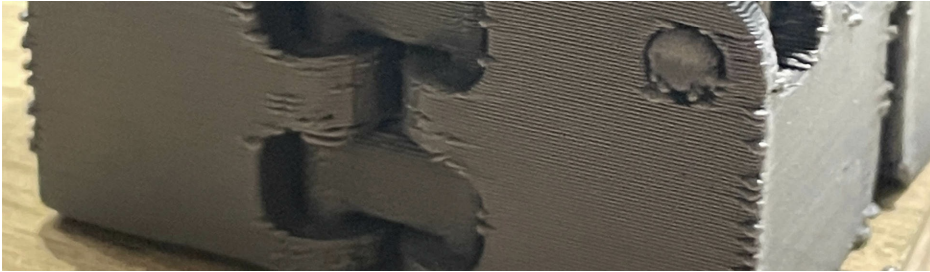
Remember to test all wires and motors before going this final route, since you don't want to spend your time and money only to have the issue occur again with the new board.

Summary of Fixes and Precautions

- If the motor is extremely hot to the touch during a print, add a heat sink. It is smart to add a heat sink to your extruder stepper regardless.
- If your motor is still overheating, add an active cooling fan.
- If overheating is an issue that a heat sink or fan cannot overcome, check the VREF of the stepper driver and make sure it isn't over your rated amount.
- Add a heat sink and active cooling fan to an overheating stepper driver.
- Check current on stepper driver if still overheating.
- If motor is malfunctioning regardless of heat, check continuity of all wires from it to the board.
- Replace or fix all wires that do not show any continuity.
- Test a different motor or plug a different driver's cords into the stepper in question to test if the stepper itself is the culprit.
- If stepper itself is not working, replace it.
- If stepper is functioning, check the driver by testing a spare.
- If driver itself is not working, replace it. If the driver is integrated, you will need a new board.
- If all above fails, purchase and install a new board.

Stringy or Blobby Prints

This chapter's failure is a result of excess material oozing out of your nozzle when no material should be extruded. Everything described in this chapter is explained elsewhere in this book, but I figured it would be smart to have all of the relevant solutions in one area.



The photo above of a blobby print was sent to me by Sen. Centrisian on Twitter.

Increase retraction settings

Retraction occurs when your printer head is travelling as to reduce any excess material coming out of the nozzle. The settings I prefer for different printer types are in both the “Materials and their Settings” chapter, as well as the “Settings Issues” chapter, but essentially you will need higher retraction settings the further your extruder is from your hotend.

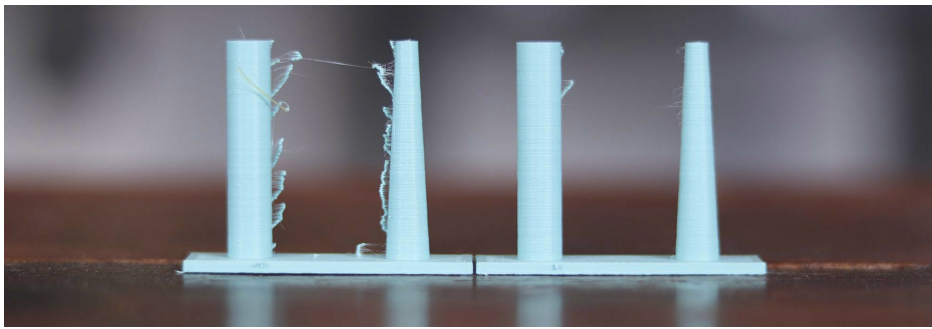
This means that when you use a Bowden setup you will need much higher retraction settings than when you use a direct extruder. You will also need higher retraction settings if there is a small gap between your direct extruder and hotend versus something like the Hemera, which has nearly eliminated that gap entirely. The smaller the gap between your extruder and hotend, the less retraction you will need, and the easier it will be to avoid blobs and strings on your print.

It is also very important to understand the material you are using. I can print something in PLA without any blobs or stringiness by just honing in my settings, but it is just about impossible to avoid some strings when printing in PETG. If you plan on printing in PETG, you will likely have to use a razor blade or a heat gun to remove the minor amount of stringing that will be inevitable.

Increase travel speeds

Since these blobs and strings will only occur when the hotend is not actively printing, you can help reduce the amount of time the nozzle is travelling by increasing the travel speeds. So long as your stepper motors and frame can handle the increased acceleration and speeds, you can bump your travel speed way up.

I currently have my travel speeds set to 200mm/s with my accelerations around 2,500mm/s/s on my CoreXY machines. You can try these numbers out on your machine, but if you experience a lot of machine rattling or hear any stepper motor skips, you will want to reduce these speeds. The highest speeds you can go without experiencing issues will drastically help reduce these strings and blobs. Since you aren't printing while the hotend is travelling, you should not experience any reduced quality in your prints with these high travel speeds.



The photo above shows two retraction test prints, the only difference between the two is an increased travel speed for the print on the right.

Play around with Coasting

Coasting replaces the last part of an extrusion path with a travel path. Replacing the last section of a print with a travel path will cause any oozed material to be used to print your part to reduce stringing. As you can see in my “Missing Layers and Holes in Prints” chapter, if you have coasting on when you do not have issues with stringing, it can result in holes on the side of your print.

That said, for the majority of inexpensive Bowden printers, you will likely see an improvement in print quality when you utilize Coasting. This is particularly true when printing in something like PETG which is more likely to cause stringing issues than PLA.

Coasting is still in the “Experimental” section on Cura, though other slicers have made it a standard setting. I expect it to move out of that “Experimental” section in coming updates to Cura.

Change Z Seam Alignment

Changing your Z Seam Alignment will help more so with blobs than stringiness. It will be essentially impossible to avoid a minor seam on your print no matter what you do, but you can mitigate its effects by changing the Z Seam Alignment.

There is a setting for Z Seam Alignment where you can choose “Random”. I do not know of a time when you would want to use a random Z Seam Alignment, but if you do choose it, you will likely see ugly little blobs all over your print. The best option for Z Seam Alignment seems to be “Sharpest Corner”, which hides these artifacts as best as your slicer can, though it will be near impossible to remove them entirely on a print that has no corners, such as a cylinder.



The photo above shows a cylinder with the Z Seam Alignment set to “Random” on the right, and the one on the left has it set to “Sharpest Corner”.

Decrease Minimum Layer Time

Some slicers have the “Minimum Layer Time” set too high for the majority of materials. This minimum layer time will cause your hotend to pause after completing a layer that finished faster than the value you have set. For instance, if I set my minimum layer time to 10 seconds, and a layer finishes in 5 seconds, your print head will pause in place for 5 seconds before starting the next layer.

This pause will inevitably lead to oozing as your hotend sits idle, and will cause both blobs and strings on your print. For the majority of materials a 3 second minimum layer time is enough, and so long as you aren't seeing curling of layers (as explored in the “Curling of Layers and Angles” chapter), you can keep yours set to 3 seconds as well.

Issues with power loss recovery

I have a new chapter in this book titled “Problems with Power Loss Recovery”, and one of the problems that can arise from this feature is blobs on your print. Some printers utilize power loss recovery by saving its progress after each layer so that you can resume where you left off in a power outage. The issue with this is that some printers take a second or so to save the progress before starting the next layer.

My FLSUN SuperRacer delta printer had this exact issue. After each layer it would pause briefly before starting the next layer, causing blobs to appear throughout the print. I had thought that I had my “Minimum Layer Time” set too high, but it ends up it was related to the power loss recovery feature on the printer. After turning off power loss recovery, the printer no longer paused after each layer and printed without any blobs.

This unfortunately means you would no longer have the capability to start a print after a power outage, but it just seems that some printer manufacturers have not implemented this feature properly on their machines.

Potential over extrusion

It is also possible that you will get some blobs on your print if you are over extruding. You can imagine if more material comes out of your nozzle than is supposed to, it can lead to not only an ugly print, but one with artifacts where the excess material squeezed out. Read the “Over and Under Extrusion” chapter to fix potential over extrusion.

Dry or Swap Materials

As with many issues covered in this book, poorly made filament or material that has absorbed moisture can also lead to increased blobs and strings on your print. Instructions on how to dry your material are covered in the “Materials and Their Settings” and the “Stripped Filament” chapters.

Summary of Fixes and Precautions

- Increase your retraction settings.
- Increase your travel speeds.
- Turn on coasting for Bowden printers.
- Try “Sharpest Corner” for your Z Seam Alignment to reduce the visibility of the seam.
- Decrease your Minimum Layer Time so that your printer isn’t paused in place when it isn’t required.
- Check to see if your printer is pausing after each layer due to its power loss recovery function.
- Check to see if you are over extruding.
- Dry or swap to new filament.

Stripped Filament

If you are noticing that your filament is stripping during the print, it may result in a part that look under extruded. This is because the shaved filament will not be fed through the extruder.

Constant filament stripping without any cleaning being done can lead to your extruder slipping, due to the teeth on your hobbed gear/bolt becoming less sharp as filament wedges itself in between the grooves. All of this will cause further under extrusion and failed prints.

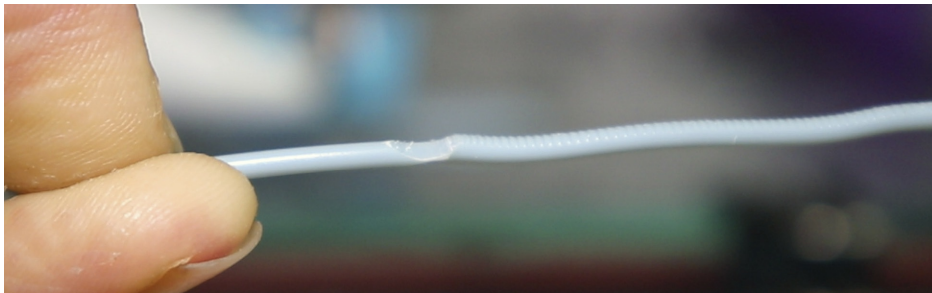
Moisture in your Material

The number one reason this problem will consistently occur is due to the material having absorbed too much moisture. When this first happened to me I did all of the steps below, but no matter what I did, the particular spool of PLA would just grind until it could no longer print. This was on a direct drive Titan extruder that I had checked multiple times for any problems.

Well, I eventually put a different spool of PLA on and it printed great. It ends up that the spool of PLA had absorbed too much moisture and could not print properly. You can notice this if you see bubbling when extruding, but another time you will notice moisture in the filament is if the material continually grinds down. This is not as common with PLA but can definitely happen, as it can with any material.

All filaments require you to store them in a low humidity area – either via vacuum sealing with desiccants, or being left with a dehumidifier set to around 25% (or just as low as it will go). You should make note of where you live, since this won't be nearly as much of a problem for those living in the desert, versus myself living in a more humid climate.

When writing the first two editions of my book I lived in Southern California where the humidity never rose very high, which meant I could be more lax with my handling of filament. Now that I live in Texas where the humidity is always higher, I need to vacuum seal every material when it's not in use.



If your filament keeps getting grinded until it looks like the photo above, you will either need to replace the spool, or get it dried out.

You can purge out the excess moisture a couple of ways. The best way to do this is to give the filament a minor vacuum purge after heating the vacuum chamber to a bit under the materials glass transition temperature. You can use your printer bed to place the vacuum chamber on as a means of heating it.

Vacuum chambers can be expensive and not everyone has one, so another method is to purchase a dry box. I own the Eibos filament dryer and it can get to 70°C, and it works great for any material. You can purchase any dry box though, but I personally recommend one that can get to 70°C. Some dry

boxes can only reach 55°C, and while those will work, they will take much longer when drying something like nylon.

If you do not wish to purchase either a vacuum chamber or a dry box, you still have a couple of methods to dry your material.

First method is just using your heated build plate and a cardboard box. Check the glass transition temperature of the particular material you are using before continuing. PLA's glass transition temperature is right around 60 degrees Celsius, so you don't want to get it hotter than that. Turn your print bed to 50-55 degrees, and then place the spool of filament on the print bed. Cover it with a cardboard box (you can use the one it was shipped in). Make sure to throw in a couple of desiccants that have not been used. Leave it there for roughly 1 hour. The heat will help to evaporate the moisture from the spool, the desiccants will help absorb said moisture, and the cardboard box will help trap the heat while doing a minor amount of absorbing as well.

It took me 3 rounds (3 hours) of this and it fixed my moisture filled spool of PLA perfectly. I was able to print as if the spool was brand new without any grinding filament.

This may not always work, so you want to make sure to use the proper precautions to avoid ever having to do this. Nylons are much more likely to absorb moisture, and thus would be a bit more difficult to complete this process. Nylon will also require a much longer amount of time when drying in this method. You may have to leave the nylon on your heated build plate for 12 or more hours.

Another method is to use your oven. This is better than the build plate, but many ovens will not go to a low enough temperature. My oven can get to 70°C but it cannot be set to 50°C. If your oven can be set to the proper temperature, you can put your spool in there for a few hours, likely over 4 hours for something like nylon. Your spool holder itself may not be able to handle those temperatures without deforming depending on what it is made out of, so keep that in mind.

Here are some guidelines for how hot you should heat common materials for proper drying: 50°C for PLA and PVA. 55°C for TPU. 60°C for ABS and ASA. 65°C for PETG. 70°C for nylon and polycarbonate. I suggest a minimum of 1 hour for PLA and a minimum of 6 hours for nylon. My dry box suggests keeping nylon in there at 70°C for a full 12 hours, so you will likely need to go longer than my minimum suggestion.

As a side note, most materials have a shelf life. Nylon material may only be good for a month or two even if you take the proper precautions. Even a year old PLA may not print as well as it was when it was first delivered to you.

Keep this in mind before you tear your hair out trying to fix a spool that you've owned and has been open for a year or longer. It may be salvageable, but it won't be that easy.

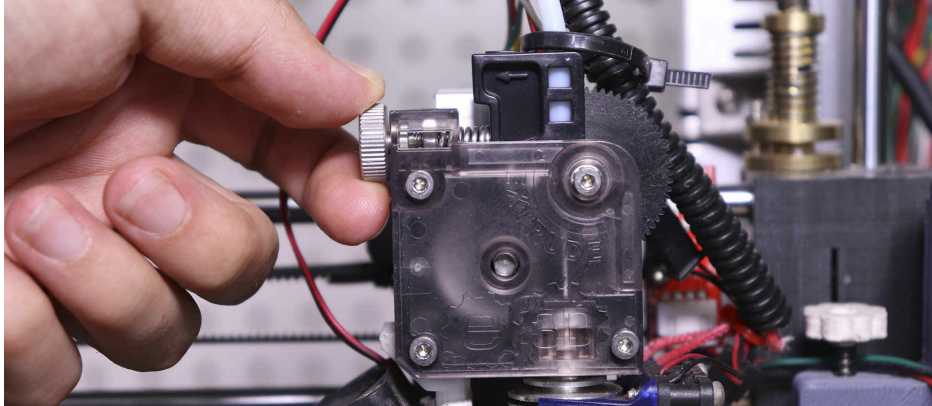
Clean your extruder

As I mention in the “Good Practices” chapter early in this book, keeping your parts clean is important to have consistent prints without failure. This definitely includes your extruder.

If you are experiencing stripped filament, or would like to prevent it as best you can, grab a small wire brush and pick and clean out the teeth on your hobbed bolt/gear.

If these grooves are not defined, they will not grip properly to your filament. This can lead to filament slipping and stripping.

Check idler tension



As mentioned for other issues in this book, you will want to make sure you have the proper tension on your extruder idler. Too tight of an idler can cause the extruder motor to skip if under powered, and too loose can cause filament grinding.

While grinded filament can occur from too tight of an idler, it will more often occur from an idler that is too loose. This is because no consistent grip is attained by your hobbled bolt/gear, and certain rotations will just rub against the filament rather than pushing it through your hotend.

It is important to get this tension right since it can cause various issues in the quality and consistency of your prints.

Increase hotend temperature

I always recommend this with caution because if you go too hot, materials are more likely to clog, causing you the need to read the “Nozzle Clogs” chapter in this book. But as mentioned in that chapter, running the hotend too cold for the speed you are feeding can lead to under extrusion.

In fact, it is smart to read the “Nozzle Clogs” section now if you haven’t already. That is because a nozzle clog will certainly lead to stripped filament, meaning if you fix the nozzle clog, you may fix the stripped filament problem.

Your extruder stepper may be turning the proper amount of steps, but the material is not getting hot enough to get to the proper viscosity. If the material does not melt properly, then it will not feed properly through the nozzle, and lead to your extruder spinning and grinding the filament.

If you are under the high end of the temperature range for your material, you can attempt printing the same G-code at a slightly higher nozzle temperature in order to help the viscosity of the material.

It is normally not recommended to go above the manufacturers temperature range, since running PLA at 240°C can give you further problems with clogging in the barrel and oxidizing material. That said, there are some exceptions, so play around with increasing your printing temperature if experiencing stripped filament.

Lower the speed/acceleration

Lowering the speed and acceleration can reduce the amount of bottlenecking caused in the hotend/nozzle.

You can imagine that if you were to speed your extruder up 10 fold you will clearly grind your filament - if it doesn't cause extruder stepper skips - since it cannot be fed through your nozzle at that speed. Your hotend needs enough time to get your material to the proper viscosity for it to feed out of the nozzle.

If you would like a tutorial on how to reduce the acceleration and speed of your print, please refer to the "Extruder Stepper Skipping" or "Settings Issues" chapter in this book.

Too small of a nozzle

As covered elsewhere in this book, using a very small diameter nozzle (0.15mm – 0.25mm) will require a geared extruder in a direct fashion. They will also cause you to lower your print speeds. This is because of bottlenecking attempting to force filament through a very small hole.

When printing with a small nozzle, if you print too fast, you will inevitably get grinded material – assuming you do not experience extruder motor skips first.

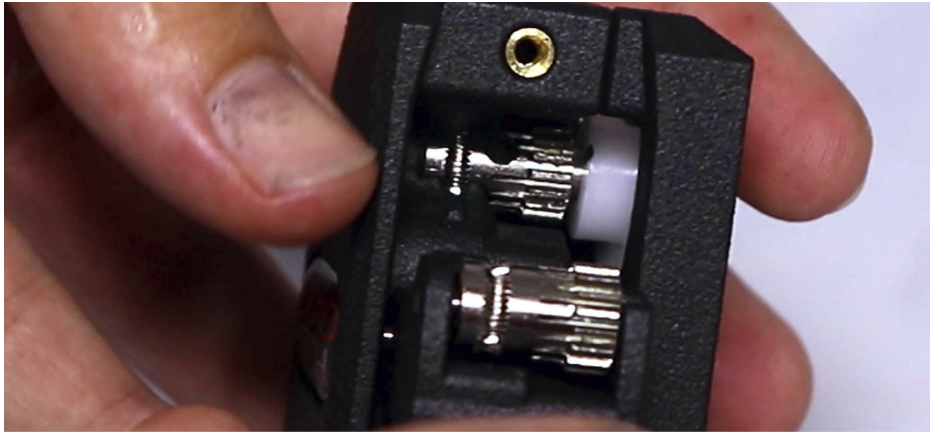
Printing Fast on Large Diameter Nozzles

You will need to slow down prints when attempting to print large layer lines on a large diameter nozzle with a standard setup. This is because the material needs a certain amount of time to heat and reach its proper viscosity to extrude. Since you are pushing so much material through the nozzle, it needs to be slowed down to prevent filament grinding.

If you will be printing on large diameter nozzles (Larger than 0.6mm) frequently, it is smart to upgrade to a hotend setup that is specifically designed to allow for high volume extrusion. E3D has their Volcano setup which is meant for just this purpose – printing fast with large diameter nozzles and layer heights. You will clearly need a geared extruder to even achieve these speeds, which is why they also sell their very popular Hemera extruder. On one of my larger machines I have a Volcano on a Hemera extruder, and it can handle large diameter nozzles with large layer heights like a pro, without the need to reduce speeds.

Replace hobbed bolt/gear

Please keep in mind that cheaply made hobbed gears and bolts may not have the proper spacing, sharpness, and depth required to grab onto the filament. These components also wear over time, particularly if you are running carbon or glass filled materials. If you are experiencing constant filament grinding regardless of the steps you took above or the material you are using, you should purchase a new extruder hobbed bolt/gear from a reputable manufacturer. Choose stainless steel options over aluminum for this part, if available.



I currently prefer Hemera extruders, but I still have plenty of Bondtech BMG's, one of which is pictured above. There are quite a few good extruder manufacturers on the market today and many of them have a gear ratio with dual drive capabilities, and some even remove as much spacing between the extruder and the heat break as possible.

Summary of Fixes and Precautions

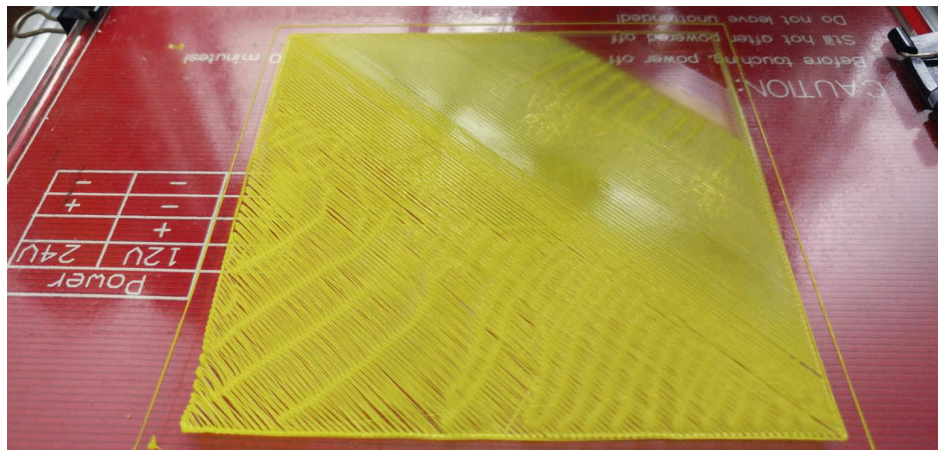
- Confirm the material you are using hasn't absorbed too much moisture. This is the most common reason for this.
- Replace filament, or dry out your material if it has absorbed moisture.
- Clean your extruder by making sure there is no filament stuck in the teeth of your extruder's hobbed bolt/gear on your extruder.
- Check to make sure you have the proper tension on your extruder idler.
- If you are below the maximum suggested printing temperature for your filament, try increasing it slightly.
- If you are using a small diameter nozzle, you will definitely have to reduce your printing speeds.
- If you are using a very large diameter nozzle, you will also need to decrease your printing speeds to get the proper viscosity, unless you upgrade to a hotend like the Volcano.
- Reduce the speed and acceleration on your print. Make sure your barrel is properly cooled.
- Replace poorly made or worn-out extruder hobbed bolts and gears.

Unlevelled Build Plate

An unlevelled build plate will mean that the nozzle is too close to the build plate in some areas and too far away in others. If the bed is extremely unlevelled, there may be structural issues with the way the build plate is mounted to the frame, however, in most cases it is minor (less than a mm height difference from one corner to another). In these situations you just need fine tune adjustments to the spring loaded corner mounts of the build platform.

Many new FDM 3D printers come with an auto bed leveling system that actually works, and there are some third party bed levelling sensors that are useful, unlike attempts in the past. I go over these a bit further in the “Upgrades and Purchasing a New Printer” chapter, as well as a later section in this chapter.

You can easily diagnose this problem by either running a bed calibration G-code, or by paying close attention to the first layer of your print. By getting used to what a print should look like on its first layer, you will easily be able to which corners are too close, and which are too far. This chapter is very similar to previous editions of this book.



(Remember that you can email me at Sean@3DPrintGeneral.com with proof of purchase for high def photos and color PDF files)

The photo above is easy to diagnose this – the front left corner has the nozzle too far from the build plate and the top right corner is way too close. Somewhere in the middle the nozzle is the proper distance.

An unlevelled build plate, along with Z-height calibration, are the two most common failures when starting a new print. Most printers either have spring loaded build plate knobs or some form of mechanical leveling system. These screws become loose over time, and even one long print can cause your bed

to become unlevelled for the next run.

If you continue to only tighten corners without adjusting your entire bed, it can result in a warped metal build plate, and eventually constant “Layer Shifts” - as mentioned in that chapter in this book.

Leveling left and right side before adjusting corner bed knobs

Before you even bother adjusting the corner bed clips, you will want to level the left and right part of your X-carriage. Variations over time, moving your printer, as well as times that you home the Z axis too close to the bed can all cause your two Z- rods to become unlevelled from each other in relation to the X carriage.

Of course this should not be needed on printers with only one leadscrew, such as the popular Ender 3. This would also only be an issue with Cartesian machines. Most CoreXY and all Delta printers should not have to worry about this.

If this is just a maintenance problem, and not an issue with your stepper motor (as mentioned in the “Steppers Motors Overheating or Malfunctioning” chapter), you can level this while your steppers are disengaged. Hold onto the left coupler that attaches the left threaded Z rod to its respective stepper motor. While holding onto the left coupler, twist the right threaded Z rod’s coupler in the correct direction to level your X carriage.



You can measure the distance from the bed to the X carriage rod on the right and left side of the build plate, and continue the above process until they are even. If you attempt to just tighten bed corners while your actual frame is off, it will result in further issues.

Loosen/tighten corner bed knobs

If you have confirmed that the left and right side of your X carriage are level to each other in relation to the build plate, and you are still experiencing an uneven first layer, you will want to play around with the corner bed knobs of your printer.

I always suggest that you loosen before you start tightening these corners. That means you should try to loosen the corners that are further from the nozzle rather than tighten those that are too close. So if one corner is too close to the nozzle, I would suggest raising the Z-height slightly and loosening the remaining 3 corners until level. This is because you do not want to be in the situation where you cannot tighten a corner any further. This can lead to a warped metal plate over time and will cause layer shifts in the Y-direction as the build plate becomes difficult to move. This is not nearly as much of a problem on CoreXY machines since the build plate only moves down in the Z-direction.

Only loosening will not always be possible though since you can imagine the above situation where 1-3 corners are too close to the nozzle and the remaining corner(s) are as loose as they can get. You can always use a longer screw and larger spring on those corners, but if that is not feasible, you will want to tighten those corners that are too close to the nozzle.

On newly built, or strong framed machines, this can be as simple as doing a slight tweak while a new print begins. Printers that have a lot of build time racked up can lead you to having a massive headache going forward with the above method. If you are unable to level the build plate easily with the above two methods, I sincerely suggest entirely starting over. If you over tighten one or multiple corners, the build plate will move with a lot of resistance, which is not good for your printer.

If you are stuck in a situation where you cannot tighten a corner further, disassemble the entire build plate. Make sure everything is assembled properly, and put it back together. Make sure that all corners have the same length screws and same exact springs and are all equally tight. Then confirm once again that the left and right side of your X carriage are level. Find the proper Z-height and then run a bed leveling G-Code that you can easily find online if not provided by your printer (just search on Thingiverse for a bed level test). Starting from scratch like this can often make this issue go away.

If you still cannot level your build plate after all of this, you likely have an issue with the frame of your printer. Take the metal bed plate entirely off and lay it on a flat surface. You can then push on each corner to see if it has

warped. If this is the case, you will need to order a new metal plate from your printer manufacturer, or at the very least, use a bed leveler.

If the plate is flat, confirm that your frame is built properly. This means that if you are using t-slotted aluminum rods, confirm that everything matches up in a 90° angle and that it lays flat on the tabletop you are using.

Confirm that all printed parts on your frame are not warped. Check to see that all bearings are popped into their holders and that everything is equal distance from each other. It would be smart to go over the “Mandatory Maintenance” chapter to make sure everything is working as it should.

If you are using an acrylic framed printer this problem will become more prevalent over time. Metal framed printers experience less warping and bending over time. Luckily there aren't really acrylic framed printers manufactured anymore.

Larger layer heights/nozzle diameter can make this process easier

If you are using a 0.25mm or smaller nozzle with 0.1mm initial layer heights or lower, you will notice extremely slight variations in how level your build plate is. As mentioned in the “Z-Height Calibration” chapter, the lower the layer height and nozzle diameter, the more difficult it is to get your first layer to stick to the build plate.

On any print with small nozzles and layer heights, I would suggest increasing the initial layer height on your slicing program. Increase the first layer to be 75% your nozzle diameter and you will have a much easier time leveling the build plate.

0.4mm nozzles can get most jobs done and will allow you to get that bed adhesion that is so hard to come by on 0.15mm – 0.3mm nozzles. On a 0.4mm nozzle I have 0.3mm layer heights for the first layer, which can save you an immense amount of headaches with a build plate that is slightly unlevelled.

A build plate on a printer using a 0.6mm nozzle with a 0.3mm initial layer height will be relatively easy to level when compared to a 0.15mm nozzle with 0.05mm initial layer height.

Please refer to the “Settings Issues” chapter for a further explanation.

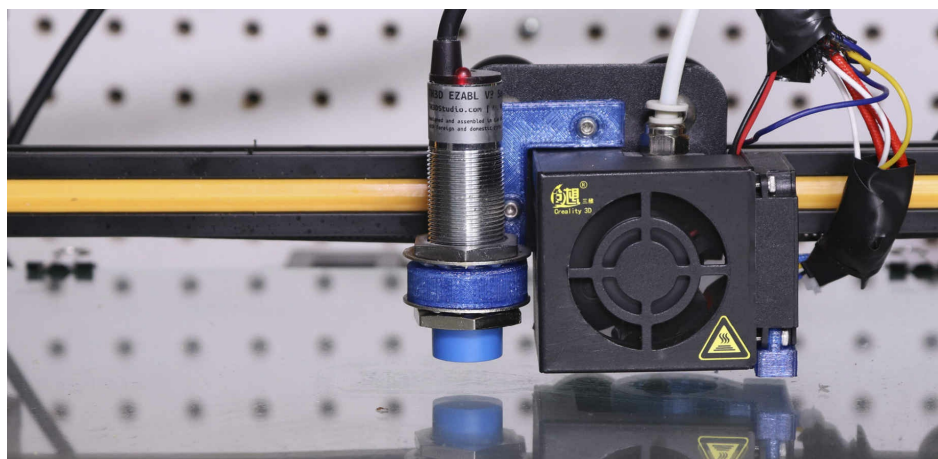
Do not frequently move your printer

FDM 3D printers are not meant to be travel ready and should stay in one spot as much as possible. Every time you move your printer, especially when traveling in a car, minor adjustments can occur that will require you to recalibrate.

The most common issue with moving your printer is having to readjust an unlevelled build plate. When we would take printers to fairs and events at my old job, we would ALWAYS have to re-level the bed when setting up. Take care not to bump into, move, or vibrate a printer, even when it is not in use.

Purchase an auto bed leveler

You can purchase a part that will allow you to auto bed level your machine. You will still need to get the proper Z-height, but your printer makes minor movements in the Z-motors to make sure the bed is level. These are technically tramming, but the industry has standardized to calling them bed levelers.



I personally upgraded my CR-10 to use TH3D's EZABL which will take a reading of the bed before printing to make sure that first layer is perfectly level. This EZABL takes 9 points on the bed in order to get a mesh, though you can adjust the amount of points it takes. After honing in the Z-offset, it then uses the Z-motor (or motors) to vary the height of the print as the nozzle moves. This means if one corner is too far from the nozzle when compared to another, the Z-motors will tweak to make sure the nozzle is closer in that section.

This minor motor movement can be seen when looking closely at the first layer. This why it is still important to get the bed as level as possible before using one of these, but it definitely helps quite a lot. I am extremely happy after upgrading to this since I no longer have to deal with this particular failure on this machine. I still have to make sure the Z-height is correct, but there is no more messing with individual corner knobs. And I promise you – an unlevelled build plate caused me far more frustrations in the past than I would like to admit.

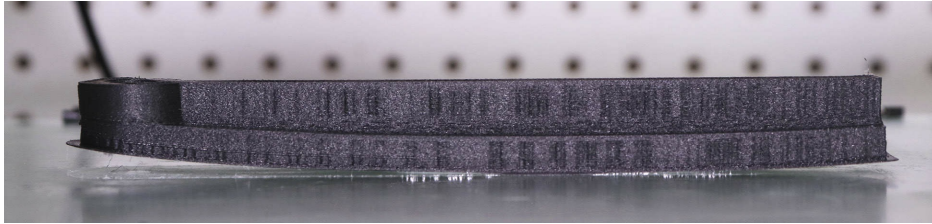
TH3D now has newer versions of their EZABL which work even better than the one I used on my CR-10, and the mechanical BL Touch has become a fan favorite among new makers. If you are worried about a warped, pitted, or unlevelled build plate, a bed leveler may be a perfect addition.

Summary of Fixes and Precautions

- Confirm the left and right side of your build plate are equally level to the X-carriage. This is only needed on Cartesian machines with dual leadscrews.
- If the left and right side are not level, while stepper motors are disengaged, hold onto the left threaded Z-rod's coupler and twist the right threaded Z-rod's coupler until level.
- Loosen the bed corner knobs that are too far from the nozzle relative to the remaining corners. Tighten only when needed.
- Start from scratch by disassembling build plate and making sure all corners are equally tightened, using same sized screws and springs with equal amount of tension.
- If still experiencing issues, confirm that your metal build plate is not warped, the printed parts on your machine are not warped, and that the frame is put together properly.
- If any part is warped or off tolerance, you will need to purchase or print replacements.
- Increase your initial layer height, or you can increase the diameter of your nozzle if you don't want to deal with this issue as frequently. A large initial layer height makes leveling the bed much simpler.
- Take care not bump into, move, or vibrate a printer - even when it is not in use.
- Upgrade to an auto bed leveler to prevent this problem from happening.

Warping

The warping of parts is just about inevitable if you don't understand the material or machine you are using. Warping is when corners or entire parts of the print curl upward due to uneven cooling, or due to improper bed adhesion. This chapter is very similar to the previous edition of this book.



Both the first and second edition of this book were written before my collaboration with PolyMaker and their “Material Science” chapter contribution, and because of this, I never fully understood why warping was occurring. Ever since working with them it has helped my understanding immensely. I can't suggest enough that everyone checks that chapter out so that you know why warping is occurring – since it may help you to diagnose and fix the problem without reading this chapter.

Understand the “Bed Adhesion” chapter, “Z- Height Calibration” chapter, and the “Unlevelled Build Plate” chapter

You need to entirely understand all three of these chapters in order to even start to try and fix your warping problems. A print will have an exponentially higher chance of warping when either part of the print, or the entirety of the print, is too far from the build plate.

This is fairly easy to understand because the further the nozzle from the buildplate, the less bed adhesion that is involved, the higher chance it will curl up later in the print.

You will need a brim on any material that has a high shrinkage rate and high internal stress rate such as ABS. For large non-circular ABS prints you will need an ABS slurry if you cannot maintain an ambient air temperature of around 45°C – all things covered in those three chapters.

I have also come to really like Magigoo’s line of bed adhesions and suggest them if you are continually struggling with warping, since I now use their products instead of making a slurry.

Understand the material you are using, and possibly use an alternative

Much of this section is covered in the “Materials and their Settings” chapter, so please review this as well when understanding your material needs.

You will almost never experience issues with warping when using a material such as PLA, because PLA has a low shrinkage rate and less internal stress (though, to understand this better, refer to the “Material Science” chapter). Very large, highly dense PLA parts should use a heated build plate and a brim, but it is very uncommon to get warping with PLA on a level build plate.

ABS is an entirely different matter, being an amorphous thermal plastic with a lot of internal stress when extruding. Since ABS also requires a higher temperature for its build plate due to its higher glass transition temperature, the differential between the bed and the ambient air is also increased.

While ABS is great for its price and functionality, this factor may make it impossible for you to achieve certain parts on your machine without warping. This is why it is important to understand the factors and features you are looking for on your print and if you can use an alternative material.

If you require mechanical functionality and affordability, but do not care about acetone vapor finishing or a high glass transition temperature, I used to suggest trying PETG. Now with Polymaker’s new line of PLA, I suggest checking out their Polymax PLA or their PLA Pro. Both of these options have very strong mechanical properties and can replace your need to use ABS, so long as heat resistance isn’t a factor.

If you require a high glass transition temperature from your material, I haven’t personally found a great alternative to ABS or ASA. One of your best options would be Carbon Fiber Reinforced ABS. This material has a great glass transition temperature of 90°C and warps far less than standard ABS, while maintaining its ability to be acetone vapor finished. The biggest drawbacks to this material is the price compared to ABS, increased degradation of your nozzle, lower layer adhesion, and a lower bend to break ratio. The problem with most materials that have a high glass transition temperature is their likelihood to warp.

Certain nylon materials that do not have a high glass transition temperature will still have a high probability of warping. This is because they are semi-crystalline with structures that take up less space when they are aligned (room temperature) than when they are chaotic (extruded). Nylon is essentially

crystalizing on your printer bed and causing warping. More information on this can be found in the “Material Science” chapter. For these nylon materials you will need to use a coat of PVA on a glass build plate, as described in the “Bed Adhesion” chapter, or even better – Magigoo’s nylon formula.

Polymaker has helped this problem of nylons with their PolyMide CoPA, which is advertised as “Warp Free”. It still has the potential to warp, but compared other nylons, it’s far less likely.

I have attempted to print Polypropylene and Acetal (Delrin) in the past, to essentially no avail. Along with requiring a high temp hotend, Polypropylene warped on just about every print larger than a cubic inch, and Acetal was impossible to get any print started without using a sheet of cardboard on top of the build plate. They just do not want to stick to standard build platforms.

Polycarbonate ABS has an amazing glass transition temperature of around 120°C but will require an enclosed environment on any print larger than a cubic inch, and an actively heated chamber for anything of decent size. This material’s warping probability is just too difficult to overcome on most machines.

For polycarbonate and polycarbonate blends, I always use Magigoo’s PC product for proper bed adhesion. I definitely suggest checking out their entire line of products if you want to mitigate warping.

Print slower and increase printing temperature

This may not work for all materials, but for ABS and ASA you can help to reduce your warping issues by extruding slower and at a higher temperature. As Polymaker covers in their “Material Science” chapter, printing slower gives the material more time to release its stress. This means that a lower extrusion speed will reduce your warping problems.

The same is true with the extrusion temperature. Increasing the extrusion temperature means more motion within the material. More motion + more time to release stress = less warping. Printing ABS as slow as possible on your machine, along with printing temperatures up to around 250-260°C, can help to reduce these internal stresses, and thus, reduce warping.

Printing in an enclosed environment

When you are printing a part on a heated build plate you are automatically working in an environment with uneven ambient temperatures. When the room is around 30°C and your heated build plate is 110°C, there is a quick change in temperature for parts close to the build plate. While internal stresses may be the biggest reason for your warping, this extreme temperature difference will also cause warping problems.

Printing in an enclosed machine allows for the ambient air to remain a bit hotter, due to the trapping of the heat given off by the build plate. This means the ambient air is closer to the glass transition temperature of ABS, allowing for more motion in the material and giving it more time to release stress. I suggest an ambient air of at least 50°C when printing in ABS or ASA.

You can purchase a printer that is enclosed, or somewhat enclosed, which works pretty great if you can afford them. You can also build a DIY enclosure with laser cut acrylic and a few printed parts. Or you can find some other build that someone has posted instructions for online. Creality now sells enclosure tents for their machines. I actually cut out holes on one of my enclosed printers and added 100W heaters to them, and my ambient air can now get to 65°C. This isn't required but it really helps me to print large parts in high warping materials without any issues.

When printing a part with a high likelihood of warping in an enclosed machine, you will want to let the bed sit at its printing temperature for around 5-10 minutes to allow the ambient air to heat up. A good ambient temperature for ABS would be 50°C, and ideal up to 65°C. You obviously would not want to print PLA in that type of environment though, since that is right around or above its glass transition temperature.

Many issues can arise when you allow ambient air to rise this high. Stepper motors and other electronics can overheat and cause your printer to malfunction. This is why you will need to have your power supply and board outside the enclosed chamber if possible, have enough heat sinks spread throughout, and keep an active fan on anything that is heating too hot. E3D actually has a water cooled option for their extruder, but it is not very common. Read up on the "Stepper Motors Overheating or Malfunction" chapter as well.

Even then you may still experience issues, so be sure to understand some basic thermal dynamics and mechanical engineering before getting your ambient air to 60°C or higher.

Make sure the build plate is not losing heat mid print

If your board is overheating or you having issues with connectivity to your heated build plate, the temperature may drop mid print. If you only watch the beginning of your print and come back when it is finished, you may not even notice this is happening other than returning to a warped part.

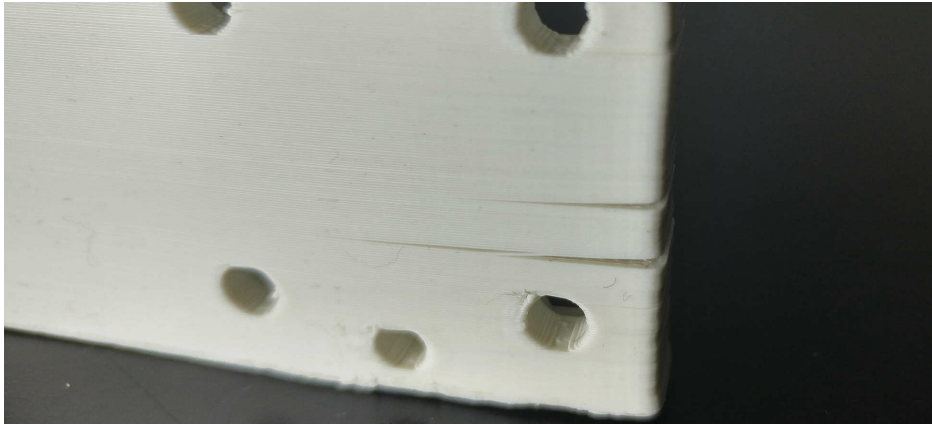
Make sure your bed is maintaining its heat throughout the entire print, and if it is not, refer to the “Build Plate Not Heating” chapter

Delamination of layers

You may not consider this failure as “warping”, but it has nearly all of the same reasons for happening. This is why it is not included in the “Poor Layer Adhesion” chapter.

If you have incredible bed adhesion, such as when you use an ABS slurry, but are printing a large part in an open environment - you can experience delamination instead of warping.

Delamination is when two layer will separate from one another, even when taking in all the layer adhesion precautions. This is because of the same temperature gradients and internal stresses explained earlier, but it occurs when bottom layers are stuck extremely well to the build plate.



The bottom of your print may not curl upward taking the entire print with it, but rather layer adhesion becomes the breaking point for this shrinkage/internal stresses.

If this is happening to you, you will need to check your slicer settings or drastically change your environment/material being used.

I have only experienced delamination on very large PLA prints when the ambient air is quite cold, while it can be unavoidable on tall ABS prints not in an enclosed environment.

Your settings can be tweaked to help prevent this delamination. The denser your part is on the inside, the more likely this will happen, so try printing your part with less infill and a couple more shell walls. Print slower and hotter to also help slow down the material releasing stresses and have more motion. You can increase your nozzle diameter to help increase the amount of entanglements between layers. Most important though is an enclosure keeping the ambient temperature high.

Finally, confirm that your E-steps are on (as explained in the “Over and Under Extrusion” chapter). If you are under extruding by a decent amount,

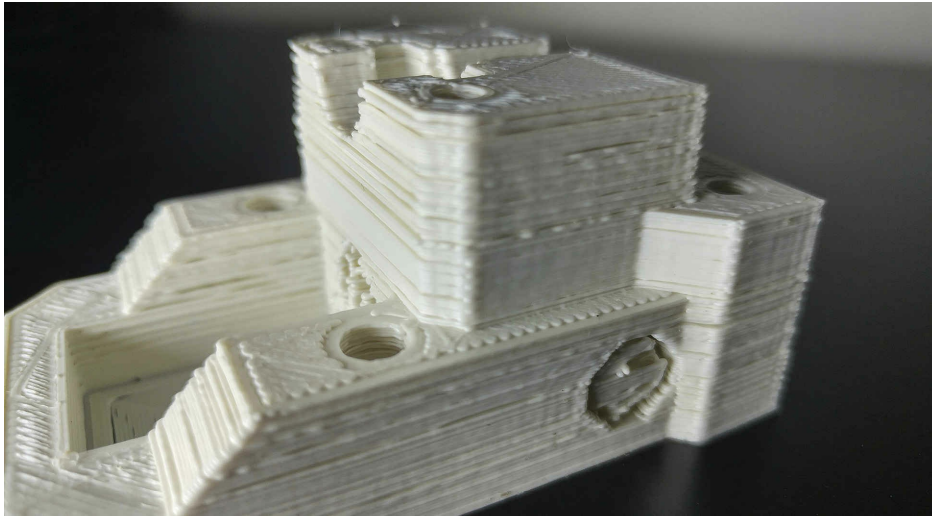
you could potentially experience more frequent occurrences of delamination.

Summary of Fixes and Precautions

- Read the “Material Sciences” chapter in this book before reading this chapter. It will help you to understand why warping is actually occurring.
- Make sure your bed is level and that your Z-height is correct when starting your print.
- Use your preferred method of bed adhesion. I have come to really like the Magigoo line of bed adhesion.
- Print slow and hot to give the material more time to release stress and increase the motion within the material.
- Print with a brim.
- Know the proper print settings for the material you are using.
- Attempt using a material with a lower shrinkage rate and internal stresses with similar applications (Polymax PLA, PLA Pro, PETG, CFR-ABS, PolyMide CoPA, etc).
- Use an enclosed 3D printer, or build an enclosure for your machine, but understand the possibility of parts overheating.
- Make sure the build plate is maintaining its temperature throughout the print.
- Delamination occurs on taller parts that have good bed adhesion. You will need to reduce the density of your part, print an enclosed environment, or use a different material in order to fix this problem.
- Print with a larger diameter nozzle for more layer entanglements.

Z-Axis Wobble

When you are experiencing Z-axis wobble, you will see what looks like repeated minor layer shifts, sometimes with every single layer. If the wobble is minor, it may just look like the print surface is not clean. You can hone in all of the slicer settings properly, but it will not fix this issue. This is an issue with the frame of your printer and not an issue with settings. This chapter is similar to the previous edition of this book.



Tighten extruder carriage and hotend setup

This is probably the most common cause for a wobbly looking print. Your extruder carriage is moving and rattling constantly over prints causing screws to loosen over time. This is especially common on 3D printed parts, since their tolerances are not as tight as mass produced carriages and extruders. This will be far less of an issue if using an E3D extruder with an E3D hotend on a \$3,000+ machine than on a \$200 stock Prusa i3 with printed parts.

This is because more expensive machines usually have higher end parts holding everything together. Lower end machines will use printed parts with minor tolerance issues, and may even use zip ties to hold belts and bearings in place. These zip ties can become stretched over time and results in a carriage that is not harnessed tight.

Frequently check to see if you can easily move or rattle your hotend/extruder setup. Lower the hotend to room temperature and literally grab the hotend and try and move it around. Don't be rough, just give it a little rattle. If you see the carriage or hotend rattle or have free play, you will need to fix that in order to get rid of this problem. Machines that don't have any free play on their hotend result in the least amount of Z-wobble.

If you can, you will likely need to tighten the screws/bolts that hold the extruder or hotend to the carriage. If tightening doesn't fix the problem, check to see if your printed parts are worn out or off tolerance. If they are, you will need to reprint them. If not, you can move on to the next solution.

Replace old/worn out bearings and make sure bearings and rollers are harnessed tight

If your carriage is rattling when you manually try to move it, but your entire extruder carriage and hotend are tightly set up, your bearings may have become loose over time. As you can imagine this is far more common with plastic bearings than with metal ones.

This bearing issue is specific to printers with linear rods, not for linear rails or aluminum extrusion with rollers. Linear rails are preferred and do not have this issue. For printers with rollers, such as those on an Ender 3, you will want to rotate the nut on the roller to hold it tight to the aluminum extrusion. You do not want your rollers to spin freely. This nut is not a “righty tighty” sort of nut. It will have more pressure on one side, and less on the other, so a full rotation will be back at the starting point. Turn this until the roller is held on tight, and do this for all axes.

When doing this rattle test on a printer with smooth rods, you should be able to see if there is a small gap in the X bearings.

I didn't think this could happen until I replaced a 2 year old printer's bearings who was experiencing consistent Z-axis wobble, and it fixed the issue instantaneously. These were on Lulzbot TAZ 5 machines that used plastic bearings. The replacement bearings had no free play and gripped the linear rods tight.

If you are using a less expensive machine that uses zip ties or something similar to hold the bearings in place, you will need to replace these with new zip ties, or find a part online to print that will hold the bearings tight. I have come to prefer linear rails just to reduce this ever becoming an issue.

Ensure your hotend and nozzle is set up properly and tight

This process is explored in the “Built up Material in Nozzle” chapter, but can also show symptoms in a Z-axis wobble. Every hotend setup needs to be assembled in a slightly different fashion, but they all require you to do final tightening with a heated hotend.

When the hotend heats, the metal expands and can cause your once tight nozzle/heater block to actually have minor gaps. This can lead to rattling of the heater block throughout the print, causing an ugly wobble in your print.

If you notice that your heater block is loose when hot, or that you constantly have to brush off the nozzle from excess material, you will likely need to tighten these parts.

I always suggest doing the final tightening of your nozzle and heater block when heated to 240°C, using proper gloves and tools. Remember that you have a high chance of burning yourself, so only do this with extreme caution. If you tighten the hotend/nozzle when at room temperature, you will find it won't be tight at 240°C.

You still want to make sure to not over tighten anything. I have broken quite a few heater blocks, nozzles, and heat break barrels due to over tightening. These parts, especially when hot, can easily snap under pressure. This hasn't happened to me in a while, but would occur when I didn't understand how careful I should be. When you are tightening your nozzle with the hotend heated, make sure to only tighten until you know that the nozzle and heat block are not loose and will not unscrew during the print – don't muscle it.

If you still are experiencing your hotend is rattling and you have made sure it's harness is tight, you will likely need to upgrade or replace your nozzle, heat block, or entire hotend. Poorly made parts will not have tight tolerances and can lead to these gaps in your threads. You can try to save money if you want, but I only purchase from reputable manufacturers for this reason. All of my hotends are from E3D, since they are one of the most respected companies in the 3D printing field.

Make sure the build plate is harnessed tight

Just as with the extruder carriage and bearings, you will need to make sure your print bed does not have any rattling in it. This will not be an issue with CoreXY machines, since the bed only moves up and down.

When using a Cartesian machine, free-play or rattling in the build plate will result in Z-wobble, just as it would with rattling in the hotend. When not printing, and with the bed at room temperature, give it a good rattle up and down and left and right. The print bed should not have any movement other than what comes from the whole machine moving. If the print bed has some free play in the bearings, rollers, or harness, this will need to be fixed.

On one of my old inexpensive DIY machines, the build plate harnesses were attached to the bearings via zip-ties. These zip ties seem to stretch over about a month or two of printing, and so I would cut them off and replace them as needed.

Just as with the carriage, you will need to replace any plastic bearings that have become worn out over time (unless using a linear rail system). For printers with rollers such as the Ender 3, you will want to rotate that nut until the rollers are holding onto the frame tight.

Finally, make sure that the parts that are connecting the bearings to the build plate are securely tightened and up to tolerance. Take your glass or other print surface off, and then tighten all of the screws that are connecting everything. These, as with all other screws, will loosen over time.

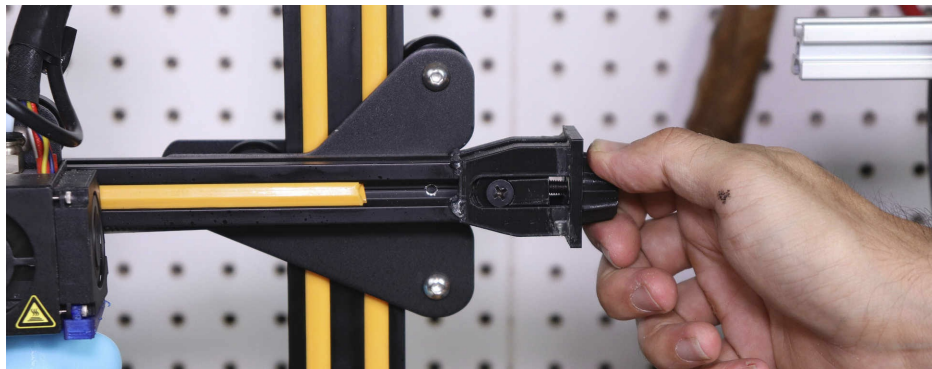
Tighten All Belts

Other than confirming all harnesses are tight and that there is zero rattling on the extruder and build plate, the next most common reason for Z-wobble is a loose belt.

As explained elsewhere in this book, it is possible to over tighten a belt, but it is pretty difficult to do so on low end machines where the belt is just held together via zip ties. Both the X and Y axis belts should be very springy to the touch with zero-droop.

If there is any droop in your belt, you will need to tighten. For low end, non-upgraded machines, cut the zip tie that is holding the belt together, grab some pliers, and pull tight as you put on a new zip tie. Make sure the belt is tighter than it was and that the zip tie is pinching everything so that the belt won't slip.

I always suggest adding a belt tensioner to printers that do not have them, and luckily the popular Ender 3 V2 has these stock. Just tighten these belts until they are springy to the touch. A loose belt will definitely lead to Z-wobble and other issues.



There is likely a file on Thingiverse for your specific machine setup if you do not have these belt tightening knobs.

Be careful when adding one of these, since you will now be able to over tighten, which I had mentioned is difficult to do without this. Just turn the knob until the belt is very springy to the touch. There is no real scientific way to do this, you just want to make sure there is zero droop whatsoever.

Check for wobbly or bent rods

Using thin threaded M5 or M6 Z-rods instead of thicker M8 or M10 leadscrews can lead to a wobbly or bent rod over time. It is a major upgrade to have thick M10 leadscrews since you will experience far less Z-wobble and will almost never see a bent rod.

If you have these thinner threaded rods, just move your printer up and down in the Z-axis. You will actually be able to see this problem easily since the rods will wobble back and forth. This is exactly what happened on my DIY machine and I fixed it by upgrading to M8 leadscrews (just make sure you print all the parts you need before disassembling and upgrading). To get a full tutorial on this, please see my old YouTube video titled “Upgrading your 3D Printer to 8mm Leadscrews”. This video is a few years old now and luckily it seems that most companies have upgraded to only using thicker leadscrews on modern machines.

There are some models on Thingiverse and elsewhere that help with this wobble in your threaded Z-axis rods, such as the “anti-wobble coupling” by toolson, and they actually work quite well. While this is true, they won’t be able to fix an actual bent rod and nothing will be quite as good as upgrading to thicker leadscrews.

A heavily bent rod is not common unless you frequently transport your machine. An actual bend in your threaded Z-axis rod will require you to replace it. I have only had to do this on one machine, one time in my history of printing, and it was on a thin threaded Z-rod.

While I have only had one fully bent rod, I have had plenty of ones that will wobble when moving up and down, leading to minor Z-wobble.

Add an anti-backlash spring loaded nut

These anti-backlash nuts are normally only meant for thicker leadscrews, and they help quite a lot to prevent any backlash when moving up and down in the Z-axis.

This is slightly confusing as to explain, but with these springs and added nuts, you can expect a lot less rattling and prints looking as if they have Z-wobble. Please take note of the pitch of your leadscrew since you will need to make sure your anti-backlash parts are the same. The majority of 8mm leadscrews have a pitch of 2, but you will just need to confirm with your printer specs (or with the part you buy online when upgrading).

Also make sure you are able to actually use these on the printer you are adding them too. I bought a set that were too thick to add to the CR-10 since their holes did not line up, which lead me having to return the set.

Lubricate guide rods and threaded rods

Your X and Y axis guide rods should be smooth enough so that both carriages can move around freely (when using a linear rod printer). Your Z-axis guide rods (the ones that are not threaded) should also be smooth enough for the carriage to move in the Z-direction without any skipping or any bearings getting stuck.

Most printers use self-lubricating bearings, but even these will require lubrication after frequent printing. If your bearings are getting stuck or having trouble moving during the print, you can experience some Z-axis wobble.



Get some white lithium grease apply with a rag to these non-threaded guide rods. Move the carriage around on all axis's so that it spreads across your bearings. This should help with the issue.

Just another reason for linear rails, they should not require any additional lubrication.

Along with the smooth rods, you should also add a bit of lubrication to the threaded rods/leadscrews. Just grab some lithium grease on a rag and rub it up and down. You want to make sure these threaded rods aren't entirely dry to the touch.

Make sure the bed can move back and forth smoothly

As mentioned in the “Unlevelled Build Plate” and “Layer Shifts” chapters, if you over tighten one or multiple corners of your build plate, you will have a lot of difficulty moving the bed back and forth (of course only on Cartesian machines). This difficulty to move may lead to stepper motor skips which cause layer shifts, but can also cause some Z-wobble. You will want to make sure your bed can move back and forth easily without a ton of friction. Follow the steps in those two chapters to help remove this problem of friction in your build plate.

Is your part too tall and skinny?

This problem will not occur quite as often on a CoreXY machine, but on Cartesian 3D printers where the build plate is being moved back and forth, a tall skinny print may end up wobbling. You can use as much brim as you like with the perfect bed adhesion, but if you are printing a tall and skinny part, it will likely wobble as the bed is moved back and forth.

This wobble will make the top of your print have this Z-wobble, while the bottom of your print looks just fine.

To be honest – there is no perfect way around this. You can manually design some anchors onto your print so that as it gets taller, it is held further in place, but just printing a very tall skinny part as is will likely result in this Z-wobble.

If I am unable to design in some further anchors, I am often forced to cut extremely thin display parts in two, to be glued together post printing.

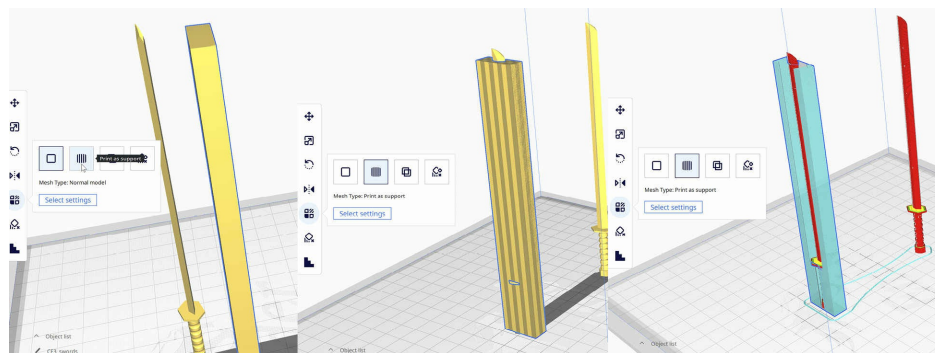
Below is a summary of how to add a very simple anchor to your part via Cura.

Anchoring Parts:

There are likely better ways to prevent rattling of tall skinny parts, but the easiest way I know of would be to anchor your print in Cura. I have rarely done this, but it definitely helps with a tall, skinny print wobbling back and forth. This is also explained in the “Parts Being Knocked Over” chapter.

Below is an example of two skinny swords from a Deadpool print that I made for my YouTube channel. When not adding any anchors, my Cartesian machine would wobble the build plate back and forth and cause the top half of these swords to look extremely ugly (if they didn't get knocked off entirely).

Cura now allows you to bring in a second model that intersects with your main print. They also allow you to print a part entirely as support. This means you can drag in a second object that acts only as support for your main structure.



This rectangle in the example above is thin, so it won't take up too much material, yet it will extend the anchoring for the sword (I added a second sword to compare how it will slice). After bringing in a shape that will work for your model, you can choose the model and click “Per Model Settings” and then “Print as Support”

After turning the shape into “Print as support”, you can then drag it over your tall, skinny print.

As you can see in “Layer Mode”, this entire shape is now support structure that can help to anchor your tall skinny print to help prevent this wobbling back and forth.

As mentioned – there are likely other ways to do this, this is just the simplest way I know of since it allows you to do this right in your slicing software.

Summary of Fixes and Precautions

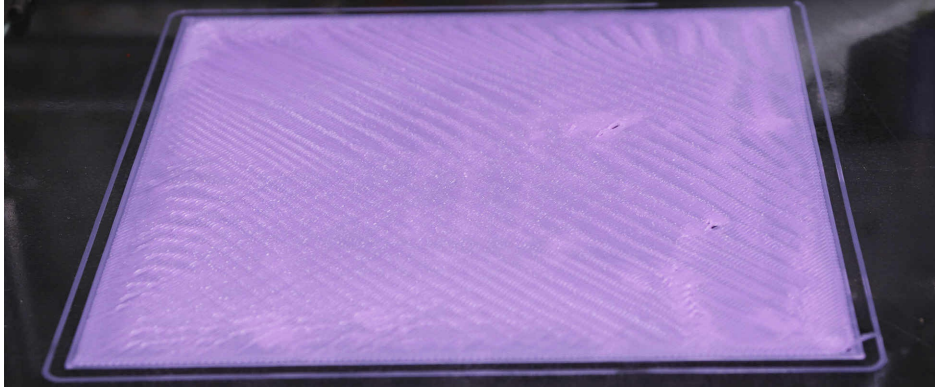
- Tighten all of the bolts and screws that connect your hotend to your printer.
- Check to see if you need to replace printed parts on your extruder carriage.
- Reduce all rattling in the X-axis carriage.
- Replace old or worn out bearings on linear rod machines.
- Tighten rollers on printers with aluminum extrusion frames (such as the Ender 3).
- Tighten your hotend setup while heated to 240°C, making sure to not over tighten.
- Remove all wobble in your print bed including replacing any bearings and tightening all harnesses.
- Tighten both the X and Y-axis belts. There should be zero droop.
- If using a thin threaded Z-axis rod, print a part that can help guide and reduce wobble.
- If possible, upgrade to a thick M8 or M10 threaded leadscrews.
- Replace any physically bent Z-axis rod.
- Lubricate all guide and threaded rods.
- Confirm the build plate can move smoothly.
- If your part is just too tall and skinny to not wobble – you may need to cut in half or manually design in anchors.
- Think about a linear rail CoreXY machine for when you want to upgrade in the future.

Z- Height Calibration

The Z-height calibration refers to honing in how far the nozzle is from the platform. This is determined on most machines by an adjustable screw that will run into an endstop. On printers such as the Ender 3, you will have to move the actual Z-endstop. Having your initial Z-height be incorrect is by far the most common mistake when starting a new print, but also the quickest to spot.

Along with many other printing failures, a combination of mechanical issues and slicer settings can be the culprit of a nozzle being too far or too close to the build plate. Even when using an auto bed leveler you will need to adjust this initial Z-height properly. This chapter is similar to the previous edition of this book.

Too Close to the Build Plate:



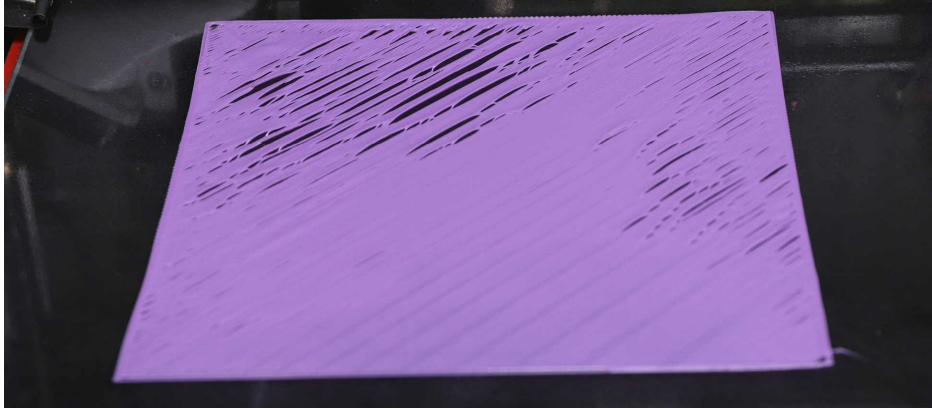
(Remember if you are having trouble seeing the images to email me at Sean@3DPrintGeneral.com with proof of purchase, and I will send over high quality photos and a color PDF)

Having a nozzle start too close to the build plate can be very problematic for your machine. Not only can it prevent material from coming out – often causing a nozzle clog and/or extruder motor stepper skips – it can actually damage your printer.

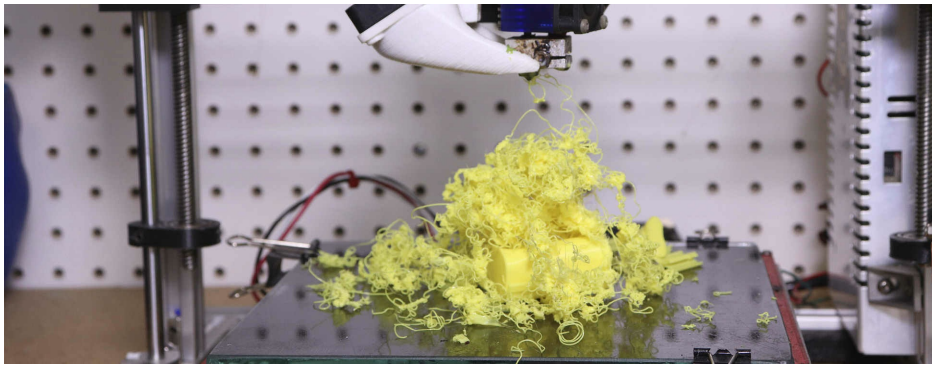
When a printer starts too close to the bed it will grind away attempting to continue printing. This can cause your printer bed to be damaged by serious scratches, and can actually degrade your nozzle. This is just another reason it is essential to watch your first layer printing before leaving your machine unattended.

Frequent printing of having your nozzle be too close can cause you to have to purchase a new nozzle and possibly a new build plate. If you are using a PEI surface, this can be an expensive fix. Never let a printer continue if you notice it is too close to the build plate.

Too Far from the Build Plate:



If you were to let your printer continue printing when the nozzle is too far from the build plate, the headaches involved can vary depending on the distance. If only slightly too far from the buildplate, your part may be more susceptible to warping and will cause gaps on the bottom layer of your print. A little further and your print may get knocked off, and very far you will be left with a spaghetti monster to clean up.



Generally, having parts start too far from the build plate will not cause as much damage to your machine, but you will be left with wasted material and a cleanup on your hands. There have been rare cases in which this mess of material engulfs the hotend as it continues to print, and once cooled leads to a hotend being submerged in a solid block of plastic. The cleanup from this can be so extreme that it may warrant or require replacing hot end components. See the end of the “Built up Material on the Nozzle” chapter for a further description of this.

Mechanically adjusting Z-height

Most machines come with an adjustable screw or lever that will run into your Z-endstop when finding home (if not using an auto bed level sensor). If your printer does not have an adjustable Z-height, I highly suggest printing one.

Files for adjustable Z-height calibrations can be found on websites such as Thingiverse, you will just have to find the correct one for your machine. The Ender 3 does not have an easy way to adjust Z-height and will require you to move the actual endstop up or down on the frame.

Being able to easily adjust the Z-height is key, because over multiple prints these mechanical machines are very susceptible to slight variations (along with just having different initial layer heights). You will want this screw to have a spring keeping pressure on it in order to prevent as many minor turns as possible throughout the rattling of the printer.

You will want your nozzle to be in a starting position where you can slide a piece of computer paper under it, but it has some resistance. This distance will be tweaked depending on the initial layer height, as mentioned later in this section. I personally am able to tell what the proper z-height via sight is since I have done so much printing, but you can always go with this piece of paper method as a starting point.

If this initial height is good for one section of your build plate, but not the rest, you will want to read the “Unlevelled Build Plate” chapter of this book.

Initial layer height

The height of your first layer will have a large correlation to how high your nozzle should be from your build plate. The smaller your initial layer height, the closer your nozzle will have to be to the build plate.

This may seem intuitive, but it is very noticeable when printing different degrees of quality. Printing in draft quality, or having your initial layer height be close to 0.4mm on a large diameter nozzle, will allow you to start prints much more simply than if you were to only be printing in very fine quality. This is because there is a lot of leeway on this thick initial layer.

This means that changing your nozzle diameter and layer height can change your Z-height calibration. Using a very fine 0.25mm nozzle will cause your printer to have to be much closer on the initial layer height than a printer with a 0.6mm diameter nozzle and a 0.4mm initial layer height.

Always max out your initial layer height based off your nozzle diameter when using small nozzles (75% the nozzle diameter). This means a 0.25mm nozzle should have an initial layer height of around 0.18, even if the rest of your print has lower layer heights. This will help you to find a proper Z-height easier.

Variations based on material/temperature

As with all materials, glass and metal expands when heated. This means that you will likely want to check your Z-height with the nozzle and bed heated to your desired temperatures. This also unfortunately means that you may have to tweak your Z-height based on what material you are printing with. Printing with ABS on a 100°C build plate will likely require a slightly different Z-height than printing with a low temp Nylon on a build plate that is 45°C.

Changing the Z-height in the “Start G-Code” by adding a Z-offset

Along with mechanically changing the Z-height, you can also change it by adding a positive or negative Z-offset to your start G-code. This is ideal for when you have specific settings based on your initial layer height, nozzle you are using, and material being extruded. You can have different machine setups for different printing parameters to have an accurate Z-height every time you print. Having a Z-offset for each material profile you have is beneficial because a heated build plate to 110°C will be closer to your nozzle than one that is heated to 50°C, due to the expansion of your bed.

I actually don't do this anymore since the auto bed leveler goes off of actual distance from the build plate, instead of an endstop that is attached to the frame. This means you would not have to do this either if you are using an auto bed leveler.

To add a unique Z-offset you need to go to the Machine Settings on Cura, but it might be elsewhere on other slicer software. Right under the section in your start G-Code of your Z axis finding home (G28 Z0), you will want to add your Z-offset with G0 Z<position> as shown below:

Start G-code

```
G90 ; absolute positioning - this line was added in case Cura
M82 ; set extruder to absolute mode - this line was added in
G28 ; home all axes
M301 H1 P18.39 I1.26 D66.91
G1 Z1.2 ; raise nozzle 1.2mm
G92 E0 ; reset extrusion distance
```

In the above example, the nozzle will raise by 1.2mm after finding home and before starting the print. So, if you found the perfect Z-height when using PLA, but you have noticed you have to raise your nozzle roughly 1.2mm every time you switch to ABS (which is pretty extreme), you could instead add this to your ABS profile when slicing parts. It is highly unlikely you will need to raise the nozzle by as much as 1.2mm, but this is just an example. Setting this in your machine settings should reduce the amount of times you have to mechanically adjust your Z-height, but will require you to create unique machine setups based on initial layer height, material being used, and nozzle diameter.

Here is a photo of what your print should look like when at the proper Z-Height:



Summary of Fixes and Precautions

- Mechanically adjust where your extruder runs into the Z-Endstop (if not using an auto bed leveler).
- Print additional parts if your printer does not have an adjustable Z-height option.
- Make sure to auto home before starting a print to make sure your nozzle is not too close or too far when starting a print.
- Increase the initial layer height in order to have an easier time honing in on the correct Z-Height.
- Small layer heights and nozzle diameters can lead to a lot of headache honing in proper Z-height.
- Recognize that different temperatures can lead to different Z-heights due to the expansion of your build plate.
- Create unique machine settings with Z-offsets in the start G-Code to reduce the amount of mechanical adjust required (if not using an auto bed leveler).

Tips if Still Not Working

While I tried my hardest for this book to be all inclusive for every printing error, there may be a unique situation not covered that you can experience, or at least one you cannot diagnose easily. Below are some good solutions if you can't fix your problem with any of the remedies described elsewhere in this book.

Turn off machine and power supply for 10 seconds and turn back on

This used to be a running joke at our facility at my old work because it was surprising how many times it fixed a problem. I would be frustrated for a half hour and someone would yell out “Did you turn it off and back on?” We would then laugh ironically as it actually worked.

For most problems it doesn't hurt to turn off your machine and power supply for 10 seconds and turn them back on.

Check frame for sturdiness and loose bolts

If you have been printing for a while without confirming all bolts on your machine are tight, or you have an acrylic frame, you can experience minor mechanical shifts resulting in ugly or failed prints.

Periodically examine your frame for any bends or loose bolts and fix or tighten as needed. Review the “Mandatory Maintenance” chapter in this book for proper precautions.

Flash firmware

This is another thing that is confusing but would work more often than I would think. Sometimes re-flashing the firmware onto a machine that was giving very strange failures would fix the problem. This isn't possible with a machine that hasn't been bootloaded (CR-10 or version 1 of the Ender 3 from the factory as an example), but the vast majority of machines should be bootloaded with Marlin to allow you to re-flash your machine. To learn how to bootload, visit TH3D's YouTube channel and website.

Newer machines such as the Ender 3V2 just require you to find the new firmware on the manufacturer's website, and then copy and paste the BIN file to your SD card. Turn your printer off, put in the SD card, and turn it on. Your printer should be flashed with the most updated software. This is a major improvement from needing to bootload, though you won't be able to tweak anything before flashing.

If you do not have access to Marlin and the original firmware, you can also do a factory reset with the "M502" command. Just remember anything you changed (such as E-steps) since you got your printer will revert to factory settings. The same is true if you were to flash with new firmware using a BIN file.

I actually do not need to do this quite as frequently as before, but it doesn't hurt to do periodically if you are experiencing hard to explain issues. It is also smart to just see if your manufacturer has new firmware since they may have fixed bugs in previous editions.

Re-slice G-Code

There is a chance your G-Code itself can be corrupted and you can fix your error just by re-slicing and exporting a new G-Code. This is covered in the “Model Errors” section.

Switch filament manufacturers

The quality of your filament matters. You could spend a full week trying out all of the solutions described in this book and still not be able to achieve a successful print if you are using subpar filament, or just old filament that has absorbed too much moisture.

I don't even bother with filaments that do not have high reviews and aren't made by well-known manufacturers, since their quality control is frequently very poor. If you are buying a spool of PLA for under \$15, you may have found a great deal, but it is also likely the company just doesn't use high quality PLA or have good quality control. Always read reviews, or check the "Resources" chapter for some of my personal favorite manufacturers.

Search online and on YouTube

If you are having a problem, it is likely someone else somewhere has experienced it. If you can search for your specific problem on Google to find a forum or a thread somewhere where someone has successfully fixed your issue. You can also search on YouTube.

I suggest subscribing to Thomas Sanladerer's YouTube channel since he covers an immense amount of 3D printing tutorials, and continually comes out with new videos. He has personally taught me a lot of what I know from these videos. His is definitely my favorite 3D printing tutorial YouTube channel. CNC Kitchen is another great one, CHEP is very informative, and I also suggest looking at Teaching Tech. There are quite a few great 3D printing YouTubers today.

If you purchased a machine from a manufacturer who makes a lot of printers, there is undoubtedly a review or two on YouTube. Just search your printer name and you will likely find them.

There is also the 3D printing Facebook group. There are currently over 97,000 members as of editing this book, and most are extremely helpful in diagnosing people's prints. I used to frequently respond to individual's problems, and love to read other's responses to help grow my own knowledge base. Be humble and state you are having trouble with a specific issue while including all relevant photos and I can guarantee you will get some good suggestions. Just don't ask them what a good machine for under \$300 is, since that has become a running joke over there. I may have self-promoted my book and my videos too much in the past on this Facebook group, so I do not want to wear out my welcome by commenting too much.

Send me an email

If you purchased this book and are still experiencing issues, feel free to shoot me an email anytime at Sean@3DPrintGeneral.com. I have helped many individuals just like yourself who just reach out to me. I may take a little while to respond though, since requests have become pretty high since this book has gained in popularity. I try to respond within a couple of business days, but that definitely doesn't always happen. I will do my best to reply as quickly as possible. The more detail and photos you have for me, the easier it will be for me to diagnose.

Resources

Below are some great resources you can use to help you in your 3D printing exploration.

YouTube Channels with Great Tutorials

The 3D Print General (My Channel)

Thomas Sanladerer

CNC Kitchen

CHEP

Breaks'n'Makes

Prusa 3D Josef Prusa

Maker's Muse

Make Anything

3D Printing Nerd

Teaching Tech

MihaiDesigns

Naomi 'SexyCyborg' Wu

TH3D Studio

ModBot

Chris Riley

Resin 3D Printing YouTube Channels

3DPrintingPro

3DPrintFarm

Uncle Jessy

Other Resources

Facebook 3D Printing Group (facebook.com/groups/makerbot)

TH3DStudio.com

IO3DP.com

Facebook group of the particular printer you are using

Guide.CTRLPew.com

Free 3D Models for Download

[Thingiverse.com](https://www.thingiverse.com)

[MyMiniFactory.com](https://www.myminifactory.com)

[Thangs.com](https://www.thangs.com)

[Cults3D.com](https://cults3d.com)

[GrabCAD.com](https://www.grabcad.com)

[Yeggi.com](https://www.yeggi.com) (compiles 3d models from many different sites, including those that cost money)

Free Software

Cura for slicing parts

PrusaSlicer for slicing parts

ideaMaker for slicing parts

Tinkercad.com for editing .stl files

MeshMixer for editing models

Autodesk Fusion 360 for editing and creating models. Just sign up as a “Start-up or Enthusiast” to not pay legally

3D Builder for Windows 10 for editing models

Repetier Host for directly connecting to printer

Octoprint for connecting multiple printers to a network

Places to Purchase Parts

3DPrintGeneral.com – where I link to many of the parts I use. I do not sell them, just link to where you can purchase them.

MatterHackers.com – my favorite for finding reputable parts

Amazon.com – be careful to not accidentally purchase off-brand from a non-reputable seller

Filastruder.com for E3D parts in the United States

E3d-online.com for Europe

Prusa3D.com

Newegg.com

DigiKey.com

Mouser.com

Reputable PLA and ABS Manufacturers That I Have Personally Used

Polymaker

Hatchbox

Proto-pasta

Prusament

eSUN

IC3D

ZYLtech

Reputable Unique Material Manufactures That I Have Personally Tried

Polymaker

3DX Tech

taulman3D

MatterHackers

NinjaTek

Proto-Pasta

Prusament

ColorFabb

Fiberlogy

Follow “Filament Frenzy” on Twitter since he is always testing new manufacturers and types of materials.

My Favorite 3D Printers (Pre-Built)

This section is a bit odd now, since I feel that a majority of very expensive machines that exist today are not really worth it, since you can either build a much less expensive machine that can do a lot more, or upgrade a less expensive machine.

\$ - Creality Ender 3V2 or Ender 5 (\$200 - \$300) - Best price for what you get, though you may want to do upgrades. Keep in mind there are an endless amount of Ender 3 clones that all may do exactly what the Ender 3 does, potentially for even less money. They do not have quite as large of a community, but they are worth looking at and at least watching a couple review videos on.

\$ - Creality Ender 3 S1 (\$300 - \$400) – This printer is brand new as of editing this book, and I just received a review unit only a couple of days before publishing. I don't believe it is even available to consumers until 2022. I have yet to do proper testing on it, but it looks like a great upgrade from the Ender 3 V2. It has dual leadscrews, a good direct extruder, magnetic buildplate, a bed leveler, and many other upgrades that make this option much better than the standard Ender 3 V2. The only thing it seems to be missing is a high temp hotend, but from my initial unboxing, it looks great. I would keep an eye out for my review of this unit after this book is published, since I am thinking I will start recommending this to anyone new to 3D printing.

\$\$ - Creality Ender 5 Plus (\$500 - \$600) - Good build volume and upgrades. I am looking forward to an Ender 5V2. This may still need an upgrade to the hotend and extruder to print in all materials consistently, but it will be a great printer once you do.

\$\$ - QIDI X-Plus (\$650 - \$700) – A printer where the build plate moves up and down, is enclosed, has dual lead screws, and has a hotend that can reach 300°C. My personal favorite in the roughly \$700 price range. It unfortunately lacks a great extruder and is not the easiest to customize or tweak.

\$\$\$ - Prusa i3 MK3S+ (\$750 - \$1000) - Highest praise from the community as a printer that just works right out of the box. I wish it was CoreXY, but no one I know of complains about their Prusa. You can save some money if you build it yourself rather than buying it pre-built. They likely have the best customer service of any printer manufacturer, which is part of what you are paying for.

\$\$\$ - This is where the community is lacking. While I just mentioned two printers from \$600-\$1000, I think there is a lot of space in the market for

more of these machines. There doesn't seem to be many printers that don't need any upgrades at a reasonable price other than the Prusa. Lulzbot has their new Sidekick, and though I haven't personally used one, I think it is a little overpriced. I hope to see a lot of printers coming out in this area in the near future.

\$\$\$\$ - Prusa XL (\$2,000 - \$3,500) – This CoreXY printer was just released by Prusa as of editing this book, and no units have been delivered yet. This machine seems like the best all-in-one printer available, at least one that is entirely pre-built. The \$2,000 version has one print head and seems like it would be the perfect edition for any print farm. You can also utilize their tool changer for up to 5 tool heads for an extra cost. I really hope that I get to test one out soon.

\$\$\$\$\$ - Ultimaker S3 or S5 (\$4000 - \$7000) – Ultimaker machines used to be a good go-to printer, but their prices haven't really reduced at all, meaning I think they are far too expensive for what they do now. They are great out of the box printers, but as you can see by the price, it is very hard for me to justify in today's market.

\$\$\$\$\$ - Raise3D line of printers (\$3,500 - \$7,300) - Large, enclosed printers. Great for real production but not necessarily for personal use due to its price. If you run a business and want to add a 3D printer, these are normally the machines companies prefer. That said, the new Prusa XL comes at a lower price point and will likely have better overall reviews.

Please note that I currently use 2 different FlashForge printers – The Creator Max 2 for dual extrusion printing and the Guider 2S. I personally think that both of these machines are slightly too expensive for what they do, and that they are all very hard to upgrade or do modifications to. That said, the Creator Max 2 is the printer I go to when I need dual extrusion and the Guider 2S is my go-to for higher temp printing. It is just difficult for me to personally suggest them when I think they are all a bit too expensive.

My personal workhorse printer is a SainSmart Coreception with an upgraded E3D Hemera extruder and Volcano hotend. This printer is no longer manufactured, but if you can find a good deal on a well-built CoreXY style printer with dual leadscrews that can easily be upgraded, I would suggest looking at it further.

There are many other printers that are great that I haven't tested, just make sure you watch review videos before purchasing.

Places to Order Prints

SD3D.com

Shapeways.com

ProtoLabs.com

PCBWay.com

Visit all3dp.com for their article on “Best Online 3D Printing Services” for a more expansive list.

About the Author



Sean Aranda began his 3D printing explorations by becoming the Operations Manager for SD3D in early 2015. Along with adding innovations to the 3D printing industry, SD3D is a production service provider for everyone from inventors to large businesses. With over a dozen FDM machines and a few hundred clients, Sean was able to gain quick, on-the-job learning.

Sean gained extensive experience with material variations and profile settings required for a successful print, since SD3D offered over 16 different types of FDM 3D printing materials while employed there.

While he had a very basic background before starting at SD3D, Sean's knowledge of printing failures grew rapidly as he helped clients get a usable 3D print within the promised timeframe. Whenever a printer failed or had a malfunction, Sean was in charge of all maintenance and repairs. He figured that he might be the perfect vessel to explain these issues and repairs in an easy to understand fashion.

Sean's three previous editions of this book, and his A-Z 3D Printing Handbook, have sold tens of thousands of copies, helping the community to achieve consistent, successful prints. This new 2022 edition has been entirely re-written to be updated for modern day 3D printing.

Since releasing the first edition of this book, Sean has been growing a 3D printing YouTube channel titled "The 3D Print General" where he goes over fun prints, in-depth tutorials, and printer reviews. In these four years of videos he has had machines printing for tens of thousands of hours to help show fun prints and tutorials and has gained over 60,000 subscribers. These videos are meant to fill in the gaps from this book and to show first hand examples of how you can fix your 3D printing failures. Many of the topics covered in this book have been covered on this YouTube channel in a 10 minute video. He has also gotten into 3D printed firearms as of late, so he also has quite a few videos testing out different builds by giving them a real

strength test.

A lot of the material in this book comes from trial-and-error over the course of two years of full time manufacturing across dozens of machines and countless hours for his YouTube channel in the four years since, as well as any updates in the field. It also includes any issues that readers of those first three books have contacted him about, but were not included in those editions.

He is open to anyone who purchases this book to send him an email at Sean@3DPrintGeneral.com with proof of purchase to receive digital copies with HD photos, and to ask any questions you may have that were not covered in this book. He may take a few business days to reply, but he will do his best to help. Your issue may be featured in the next edition!